



ADAPT-ABI

Construction Phase Modeling and Analysis

(Basic Module)

This supplemental reference manual is made available to users of ADAPT-ABI 2012 to help them understand the underlying modeling and analysis capabilities of the software. It references the previous, text-based INP file format used to define models. The current version of ABI uses a similar INP file format to send model information to the analysis engine.

Copyright 1997-2012

LIST OF CONTENTS

Chapter 1 OVERVIEW

1.1 GENERAL.....	1
1.1.1 Program Development.....	1
1.1.2 Segmental Construction	1
1.2 SCOPE.....	10
1.3 DISCLAIMER.....	11
1.4 PRINCIPAL STEPS IN DESIGN/ANALYSIS OF SEGMENTAL CANTILEVER BRIDGES	11
1.4.1 Construction Phase.....	11
1.4.2 Completed Structure.....	12

Chapter 2 PROGRAM DESCRIPTION

2.1 OVERVIEW	1
2.1.1 Structural Model.....	1
2.1.2 Principal Assumptions.....	4
2.1.3 Output	6
2.1.4 Solution	6
A. Node operations	6
B. Addition of new frame elements	7
C. Solution scheme	7
2.2 GEOMETRY.....	9
2.2.1 Overall Frame Geometry	9
2.2.2 Change in Section Along Frame Line.....	9
2.2.3 Flexibility in Selection of Cross-Section Geometry	9
2.2.4 Boundary Conditions.....	10
2.3 MATERIAL PROPERTIES	14
2.3.1 Concrete	14
2.3.2 Nonprestressed Steel.....	14
2.3.3 Prestressing Steel	15
2.4 PRESTRESSING.....	15
2.4.1 Tendon Number and Stressing.....	16
2.4.2 Tendon Geometry	16
2.4.3 External Tendons; Unbonded Tendons.....	16
2.5 LOADING.....	18
2.5.1 External Forces	18
2.5.2 External Displacements	18
2.5.3 Temperature Changes	18
2.6 TRAVELER AND FORMWORK OPERATIONS.....	19
2.7 CONSTRUCTION PHASE.....	21

Chapter 3 USER'S MANUAL

3.1 OVERVIEW	1
3.1.1 Capabilities	1
3.1.2 Preparation of Input Data	1
3.2 COMMAND SUMMARY.....	2
3.2.1 Natural Sequence of Commands	2
3.2.2 Summary of Commands	3
3.3 SAMPLE INPUT FOR A NONPRESTRESSED CANTILEVER BEAM.....	5
3.3.1 Description of the Structure.....	5

3.3.2 Input File	6
3.4 SAMPLE INPUT FOR A PRESTRESSED CONCRETE BOX GIRDER BRIDGE.....	7
3.4.1 Description of the Structure.....	7
3.4.2 Input File	8
3.5 DETAILED DESCRIPTION OF COMMAND SYNTAX.....	10
ACTIVATE EXTRACT	10
BUILD.....	10
CAMBER	11
CHANGE COMPLETE	12
CHANGE STRUCTURE.....	12
CONCRETE PARAMETERS.....	13
CONCRETE PROPERTIES	23
DE-STRESS.....	25
ELEMENTS.....	26
FRAME	26
LOADING	31
MESH COMPLETE.....	35
MESH INPUT	35
MILD STEEL PROPERTIES	35
MOVE.....	36
NODES	37
OFFSET DATA	38
OUTPUT	39
PRESTRESSING STEEL.....	41
REMOVE.....	42
RESTRAINTS.....	42
SECTION PROPERTIES.....	45
SEQUENCE.....	47
SET	49
SOLVE	50
SPRINGS	51
START	52
STAY ANALYSIS	53
STOP.....	54
STRESS	54
TENDON GEOMETRY.....	56
TITLE.....	62
TRAVELERS.....	62
UNITS.....	62

Chapter 4 **THEORY**

4.1 FRAME ELEMENT FORMULATION.....	3
4.1.1 Assumptions and Scope	3
4.1.2 Geometry of Internal Displacement Fields.....	3
4.2 INTERNAL DEGREES OF FREEDOM.....	7
4.3 MATERIAL CONSTITUTIVE RELATIONSHIPS	8
4.3.1 General Assumptions.....	8
4.3.2 Concrete Component	8
4.3.3 Nonprestressed Steel Component	11
4.4 PRESTRESSING TENDON FORMULATION	12
4.4.1 Tendon Discretization.....	12
4.4.2 Tendon Geometry Definition	12
4.4.3 Determination of Initial Tendon Forces	17

A. Friction losses	17
B. Anchor slip (seating) losses	19
C. Influence of stressing procedure	20
D. Discretized segment forces	20
E. Equivalent prestressing loads	22
4.4.4 Material Constitutive Relationships	22
4.4.5 Stiffness and Load Computation	26
A. Segment stiffness matrix	27
B. Segment internal force due to nodal displacements	27
C. Equivalent nodal loads due to initial forces	28
4.5 TRAVELING FORMWORK FORMULATION	28
4.5.1 Traveler Function and Purpose	28
4.5.2 Traveler Geometry	29
4.5.3 Material Constitutive Relationship	29
4.5.4 Stiffness and Load Computation	30

Chapter 5 EXAMPLES

5.1 CANTILEVER CONSTRUCTION BOX GIRDER BRIDGE EXAMPLE	3
5.2 BRIDGE PARTICULARS	3
5.3 ANALYTICAL MODEL	9
5.4 LOADINGS & CONSTRUCTION SEQUENCE FOR THE TIME DEPENDENT ANALYSIS	12
5.5 COMPUTER INPUT	14
5.6 COMPARISON OF RESULTS	24
5.7 AMERICAN, SI AND MKS SYSTEM OF UNITS EXAMPLE	26
5.8 STRUCTURE	26
5.9 RESULTS	27

Appendix A OUTPUT EXAMPLES

A.1 OUTPUT ORGANIZATION	3
A.1.1 Output Overview	3
A.2 EXAMPLE	11
A.3 INPUT FILE	12
A.4 OUTPUT LISTING	13
A.5 GRAPHICAL DISPLAY OF OUTPUT	19
A.6 INPUT/OUTPUT GRAPHICAL DISPLAY	21

Appendix B MODELING OF CONCRETE'S TIME DEPENDENT BEHAVIOR

B.1 STRAIN COMPONENTS AND THE SUPERPOSITION METHOD	4
B.1.1 Mechanical Strain	5
B.1.2 Aging Strain	6
B.1.3 Creep Strain	6
B.1.4 Shrinkage Strain	6
B.1.5 Temperature Strain	8
B.2 PREDICTION OF TIME DEPENDENT MATERIAL PROPERTIES	8
B.2.1 ACI Committee 209 Recommendations	8
A. Strength and stiffness	8
B. Creep strain	9
C. Shrinkage strain	10
B.2.2 CEB - FIP Committee Recommendations	11
A. Strength, modulus of elasticity and aging of concrete	11
B. Creep strain	13

C. Shrinkage strain	15
B.3 MATHEMATICAL MODELING OF THE CREEP STRAIN COMPONENT.....	15
B.3.1 Theoretical Background.....	15
B.3.2 Calculation of the Creep Strain Increment.....	18
A. Constant stress and constant material parameters	18
B. Linear stress and constant material parameters	19
C. Linear stress and linear material parameters	20
B.3.3 Determination of creep compliance coefficients	21

Appendix C REFERENCES

Appendix D BACKGROUND CAMBER CALCULATION

D.1 INTRODUCTION.....	3
D.2 NUMERICAL EXAMPLE.....	5
D.2.1 Input data for ADAPT-ABI	9

LIST OF CONTENTS

1.1	GENERAL	1
1.1.1	PROGRAM DEVELOPMENT	1
1.1.2	SEGMENTAL CONSTRUCTION.....	1
1.2	SCOPE	10
1.3	DISCLAIMER	11
1.4	PRINCIPAL STEPS IN DESIGN/ANALYSIS OF SEGMENTAL BRIDGES AND FRAMES	11
1.4.1	CONSTRUCTION PHASE	11
1.4.2	COMPLETED STRUCTURE.	12

This Page Left Intentionally

BLANK

1.1 GENERAL

1.1.1 Program Development

ADAPT ABI, software is a PC-based computer program for the time-dependent and load-history analysis of prestressed, post-tensioned, and non-prestressed concrete bridges and frames, in particular bridges of segmentally erected cantilever construction. It is based on the wealth of over three decades of research, bridge design practice and construction technology in USA - with due recognition that many of the adopted construction technologies were originated in Europe.

Professor Alex C Scordelis, and his supervisees at University of California - Berkeley (UCB) spearheaded a series of concerted research projects since 1960s, aimed at understanding the behavior of concrete bridges. Many of these projects concluded with a companion analytical computer software in addition to the project's research report. The works, which in many cases were supported by the National Science Foundation or the California Department of Transportation, have provided a solid basis for the clear understanding of concrete bridge behavior. Several of the research projects account for the time-dependent response of concrete under loading and environmental effects, and are particularly applicable to segmentally constructed bridges. The projects under professor Scordelis have led to a series of frame software. Researchers with significant contribution to the related work in alphabetical order are: A. F. Kabir, M. A. Ketchum, A. R. Mari, M. A. Mukaddam, S.F. Van Zyl, and Y. J. Kang. Refer to Appendix C for references.

ADAPT-ABI draws upon the pool of information generated in past research and practice. It compiles, modifies, and expounds the reported research with the objective to meet the demands of consulting engineers engaged in concrete bridge and prestressed frame design.

It is important to be clear at the onset regarding the basis of the work. The names and agencies mentioned in the foregoing are purely for acknowledgment and recognition of their valuable works in, and their contributions to the field of analysis of concrete bridges. It is neither intended to draw credence for the work presented herein, nor to shed any liability on the persons and agencies named. ADAPT-ABI is an independent entity.

1.1.2 Segmental Construction

A segmentally constructed bridge or frame, is built from discrete components which are assembled to form the complete structure. The assembly generally takes place over a period of time. In addition, a segmentally constructed bridge has one or more of the features described below:

- (i) The components are called upon to carry loading in a configuration and through a construction-phase structural system other than that of the completed structure. A good example is balanced cantilever construction, where

- the cantilevering structural system of the bridge is for construction phase only (**Fig. 1.1.2-1(a) and (b)**).
- (ii) The construction loading on the bridge components, either through construction equipment, or assembly, results in stresses which exceed those of the completed structure. Hence the design of the components, including the prestressing, is in part controlled by the construction technology. Prior to placing the closure segment the selfweight moment at the tip of the cantilever is zero (**Fig. 1.1.2-2**). The installation of the continuity tendons (for clarity shown in **Fig. 1.1.2-2(b)** below the superstructure) induces a moment which reduces the moment due to selfweight at completion of the structure (**1.1.2-2(c)**).
 - (iii) Weight, and depending on the form of deployment of the erection equipment, the stiffness of the construction equipment as shown in **Fig. 1.1.2-3** for the form traveler results in notable stresses and impacts the deformation of the bridge during the construction.
 - (iv) Early-age loading of concrete, oftentimes within the first 24 to 48 hours after casting, leads to high deformation values which must be carefully evaluated and accounted for deflection and camber control of the completed structure. High creep values and low modulus of elasticity of young concrete in leading segments of balanced cantilever construction result in larger displacements than normal under the weight of newly cast segments.
 - (v) The method of construction greatly influences the initial stress in the completed structure to the extent that the analysis of the completed structure without regard to its construction scheme becomes irrelevant. Consider classical case of the span-by-span construction of a two span bridge made continuous over the common support (**Fig. 1.1.2-4**). At completion of the bridge, the selfweight moment at interior support is primarily governed by the method of construction. The distribution follows that of the preconstruction simply supported girders. The moment due to live loading, however, is distributed following the structural system of a two-span continuous girder (**Fig. 1.1.2-4(d)**).
 - (vi) The bridge undergoes significant changes in its load carrying structural system during its erection. Spliced-precast-prestressed girder bridges are generally assembled with interim supports. As simply supported members they carry their self-weight. When spliced they take the load of freshly placed topping before the composite action sets in (**Fig. 1.1.2-5**). Another example is the incrementally launched bridge (**Fig. 1.1.2-6**), where each section of the bridge will be subject to both the negative moment associated with the support and the positive moment associated with the midspan location.
 - (vii) Geometry control during the construction is a central consideration in achieving the design dictated profile of the completed bridge. Consider the balanced cantilever shown in **Fig. 1.1.2-7a**. If built with no consideration to geometry control, the finished structure is likely to have a deflected profile marked A. However, if, as in common construction practice, it be desired that for either alignment considerations or otherwise the tip of the constructed cantilever be at a point such as D, depending on whether the cantilever is constructed using

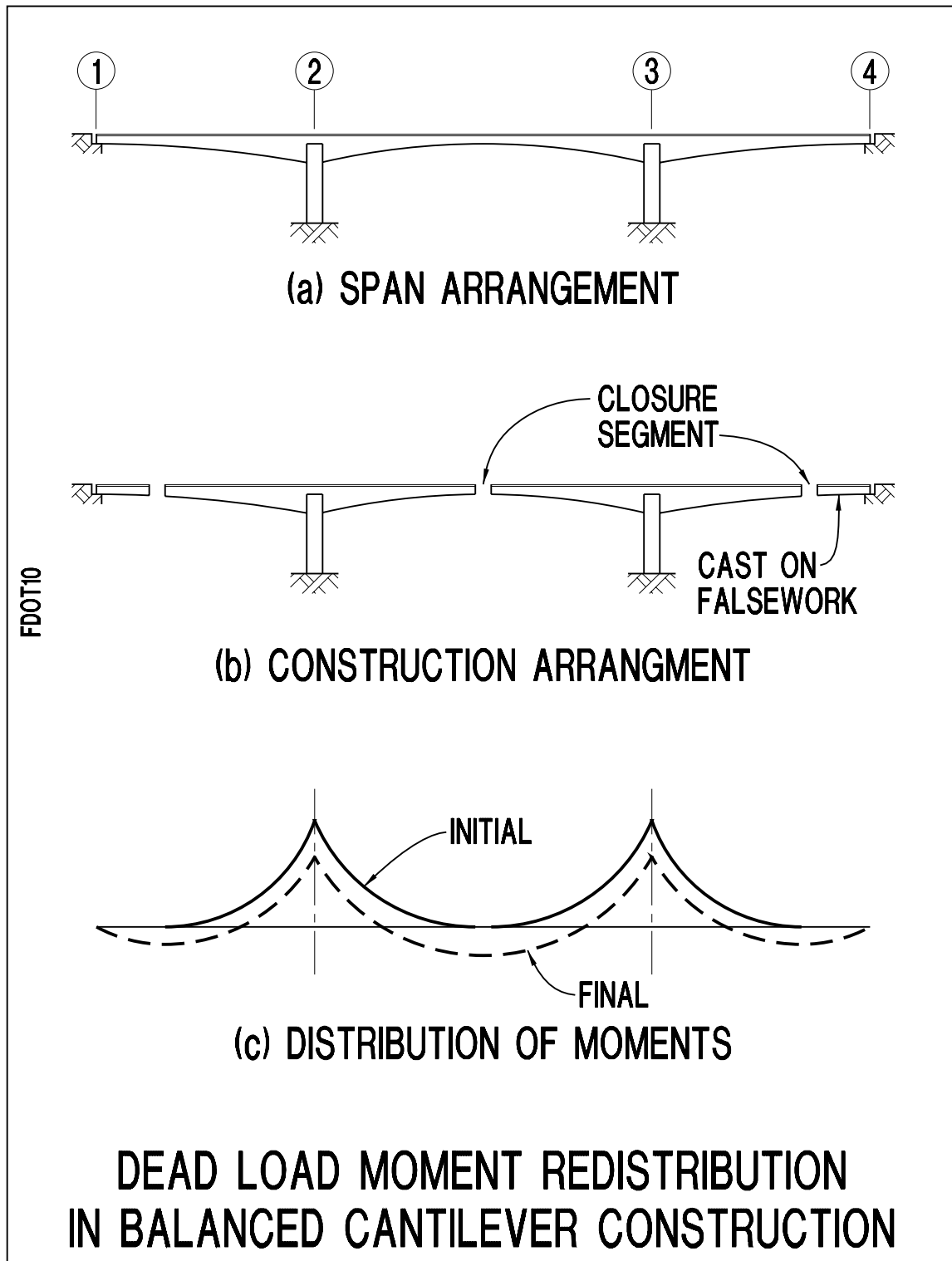


FIGURE 1.1.2-1

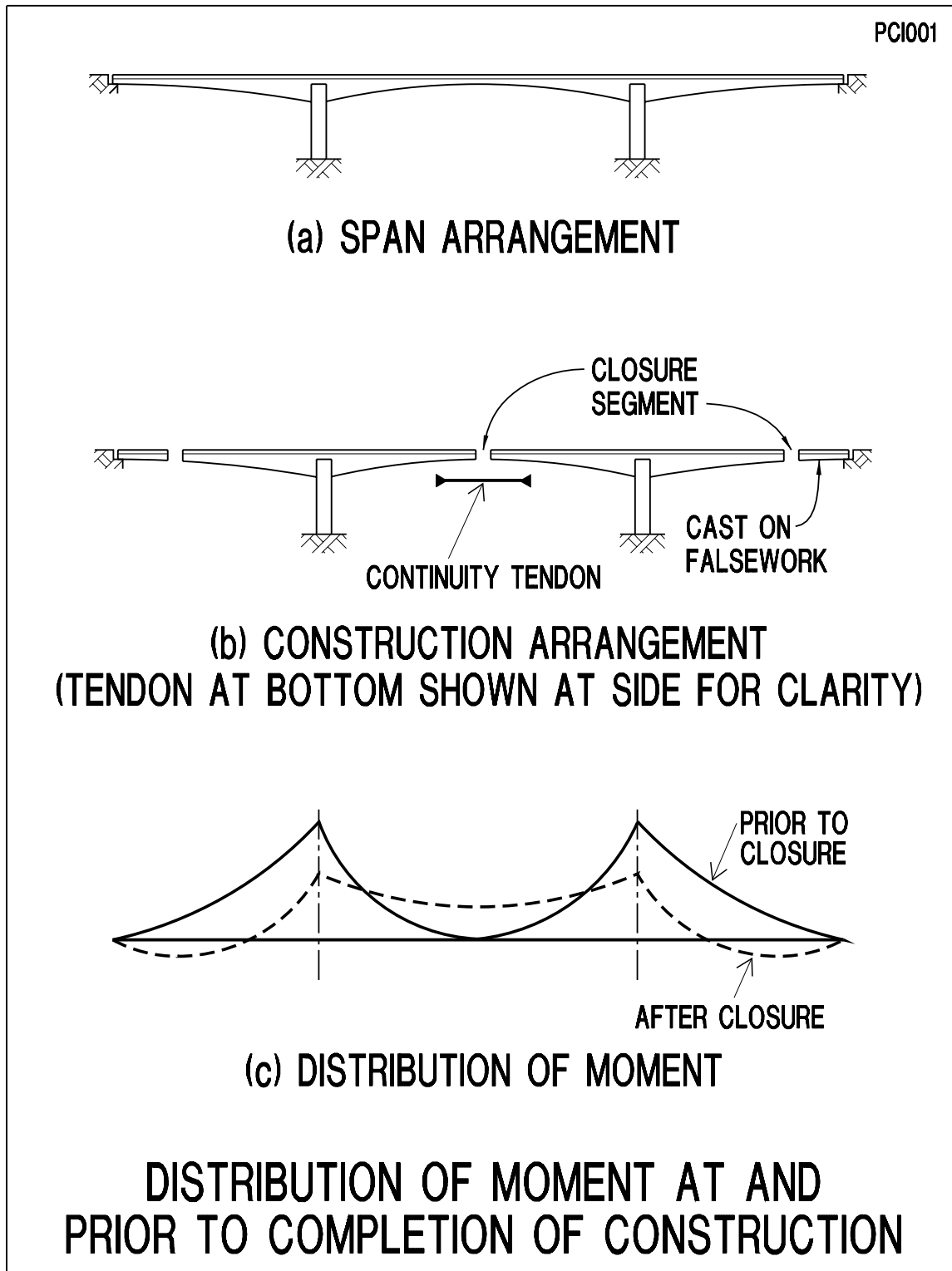


FIGURE 1.1.2-2

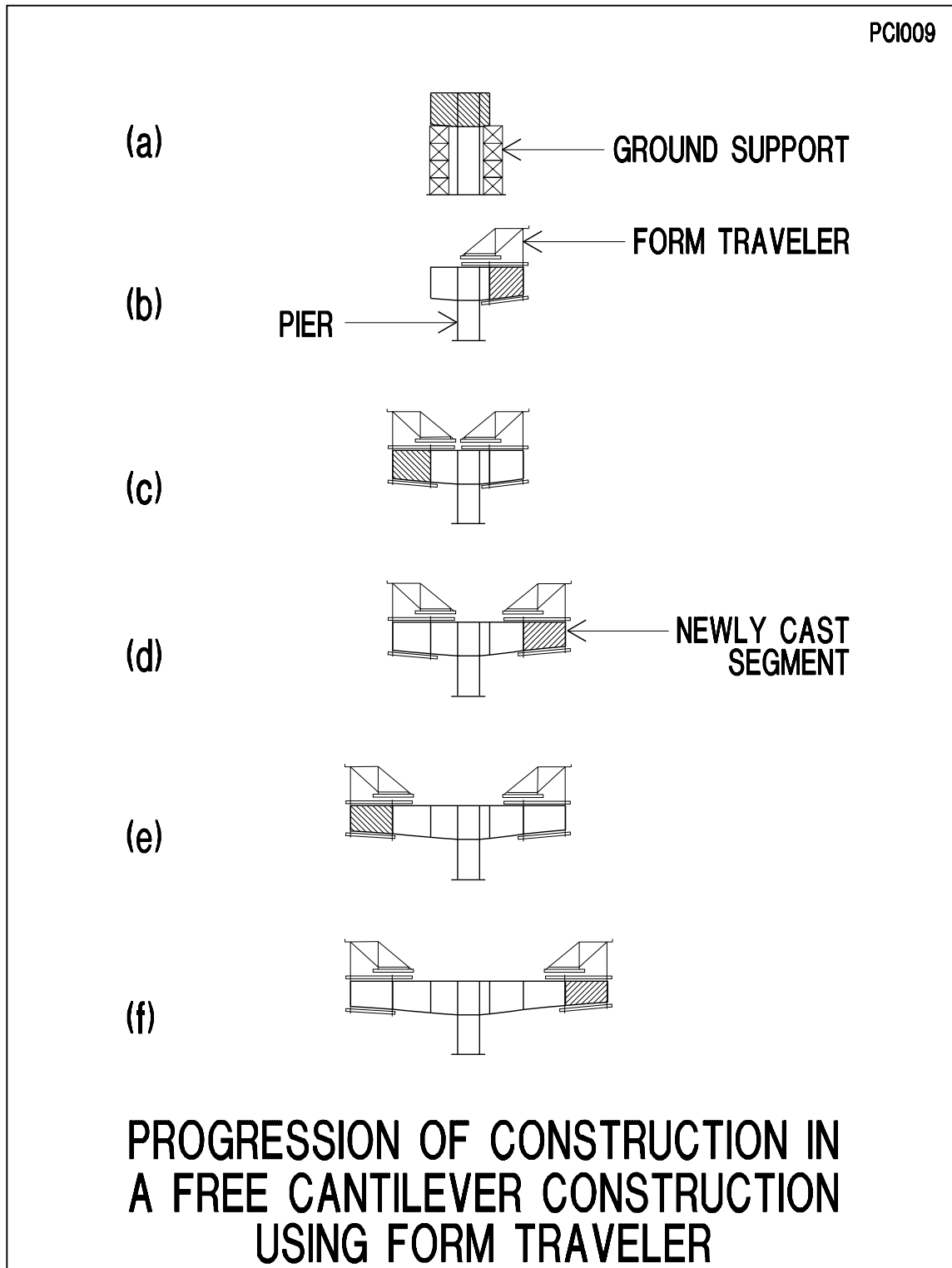


FIGURE 1.1.2-3

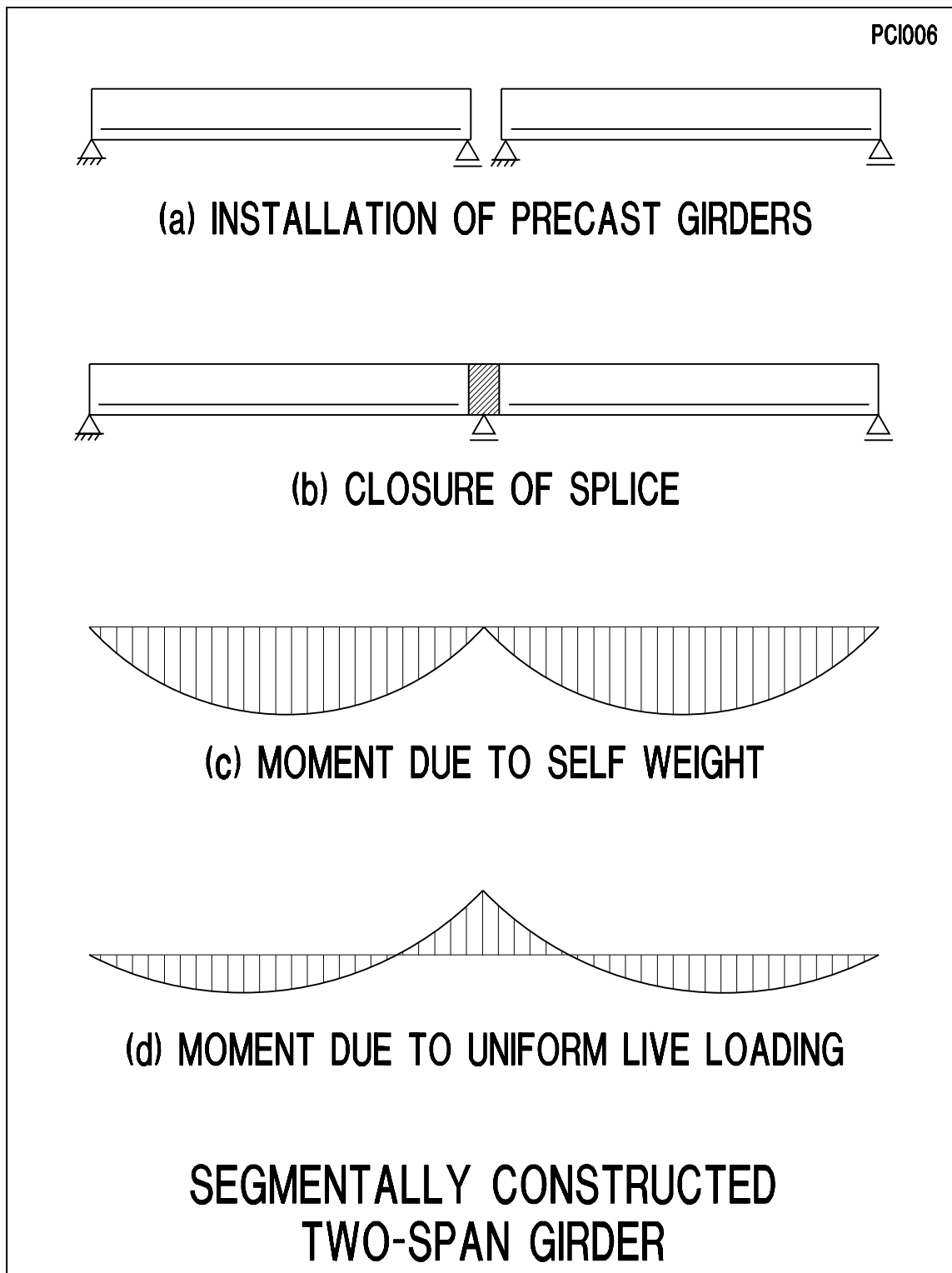


FIGURE 1.1.2-4

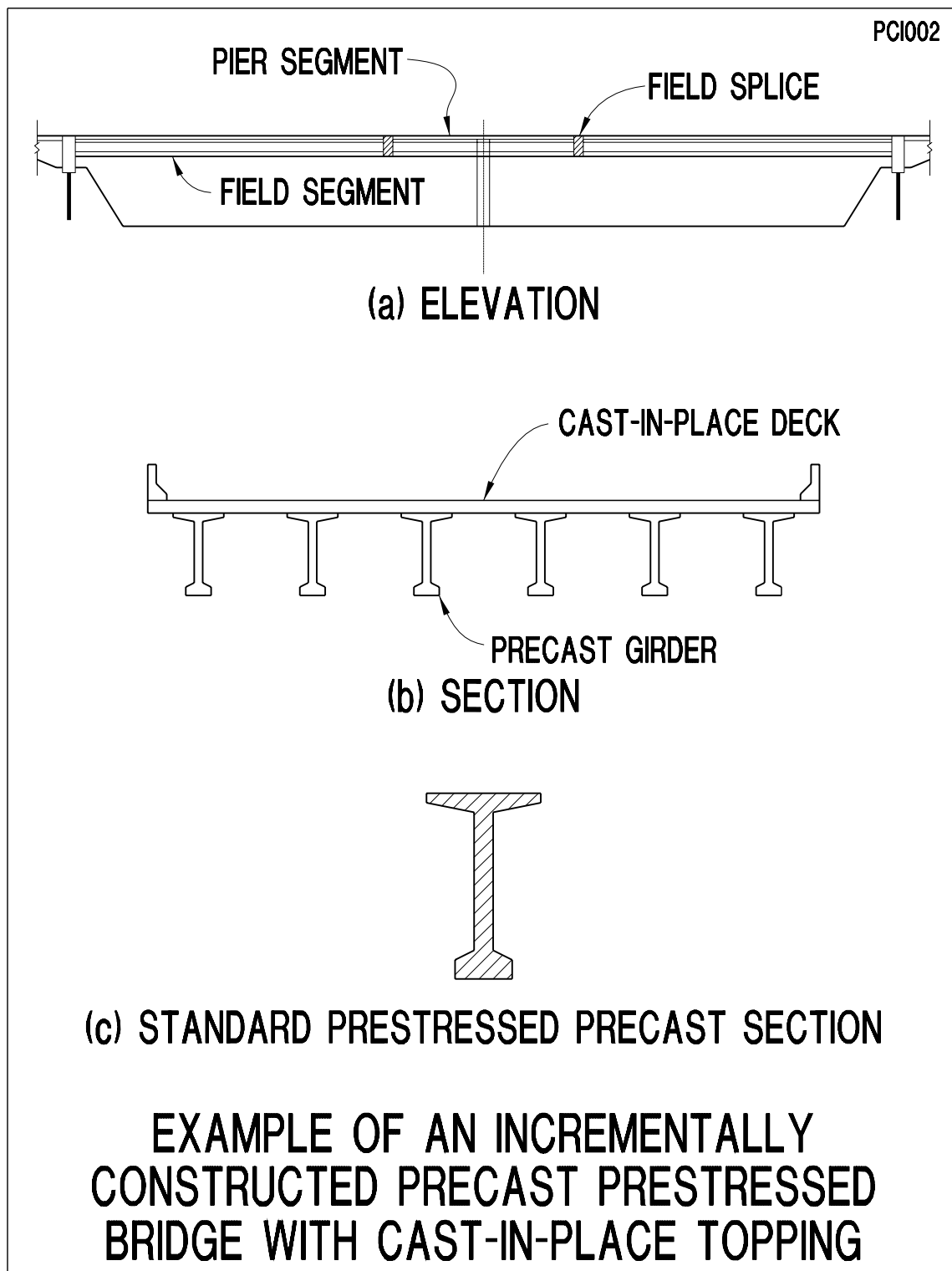


FIGURE 1.1.2-5

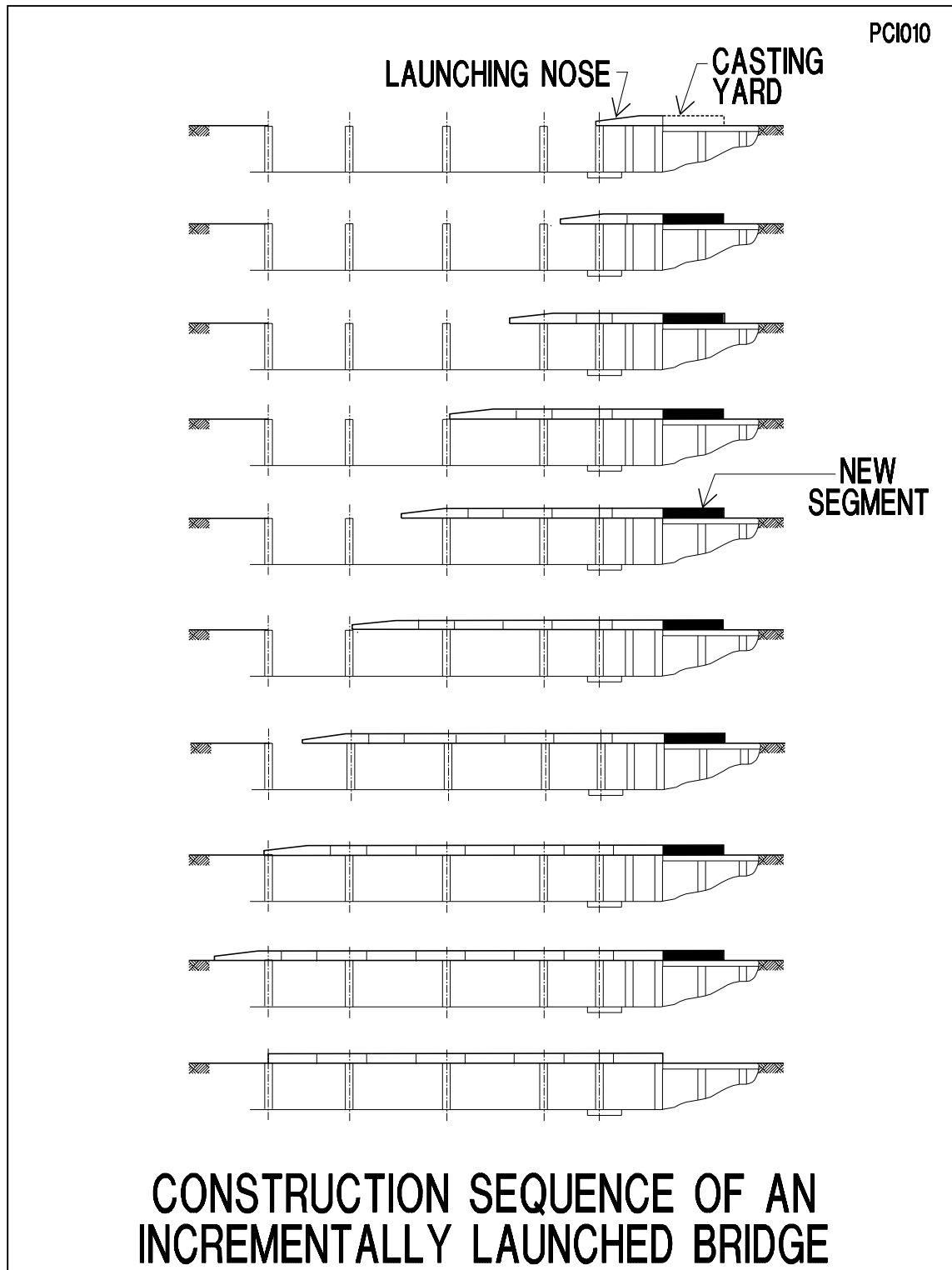


FIGURE 1.1.2-6

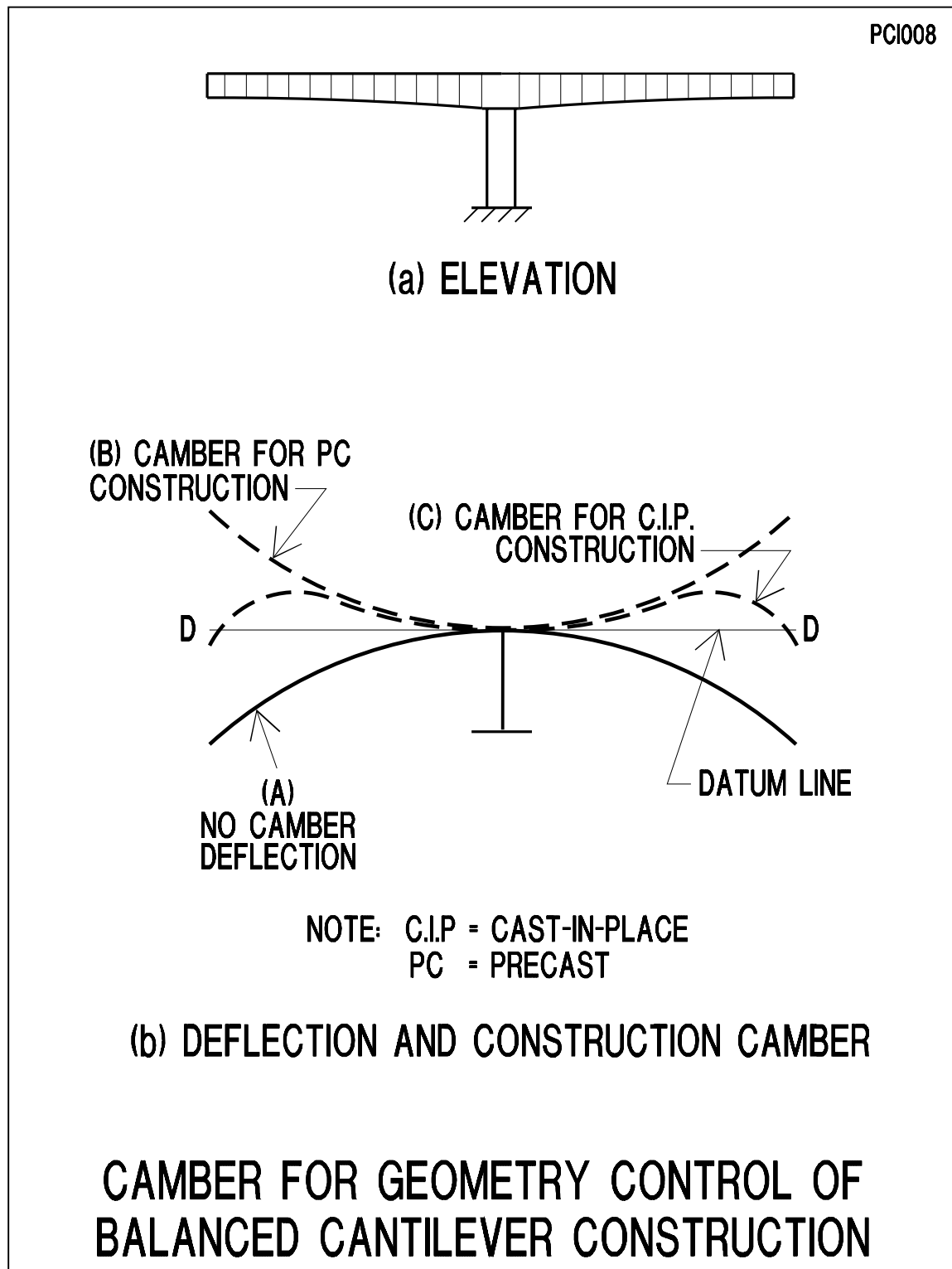


FIGURE 1.1.2-7

a cast-in-place scheme or precast segments, the construction should have a camber such as shown schematically with profiles C or B. The camber curves in the figure signify where at construction time a given segment should be placed, so that at the time determined by the designer the bridge superstructure be along the datum line (line D-D).

- (viii) Most retrofit projects involve addition of fresh concrete, external or internal prestressing, and recently developed synthetic fabrics (wrapping). Mixed material properties, the interaction of shrinkage and creep strains of the freshly placed concrete with the retrofitted components in resisting the applied loading, and the subsequent redistribution of loading among the new and existing components require time-delayed analysis specific to segmental construction.

In summary, where the effects of time, changes in the structural system, and high construction loads impact the performance and safety of a structure, during construction and when complete, and where the geometry of the completed structure depends on the method of construction, there is need for a segmental construction analysis procedure.

Prime examples of segmentally constructed bridges are:

- Prestressed-precast-spliced girders with post-tensioning and concrete topping;
- Balanced cantilever construction;
- Various schemes of span-by-span construction for continuous bridge frames;
- Incrementally launched bridges;
- Bridges retrofitted with external tendons, concrete jackets, and synthetic fabrics;
- Cable stayed bridges; and,
- Suspension bridges.

In the traditional non-segmental design, the design engineer assumes the structure in its final configuration. The method of construction is disregarded. Selfweight, and other loading are applied to the completed structure in order to calculate the design actions (moments, shears and axial loading) and deformations. It is generally assumed that there are no locked in (initial) stresses in the completed structure, and that the profile of the completed structure is as shown on the structural drawings. The design assumptions made for the traditional structures can not be extended to segmental construction. The parameters which impact the safety and performance of a segmental construction and need be accounted for in design may be grouped into time-dependent parameters and construction schedule and sequence.

1.2 SCOPE

ADAPT-ABI software can analyze both the completed bridge structures and bridges during construction, where construction procedure and sequence affect the performance of the completed project.

In addition to capabilities of other software in determining actions and displacements of prestressed concrete bridges, ADAPT-ABI has the added capability of accounting for the time dependent effects of creep, shrinkage, aging of concrete, loss in prestressing, temperature, settlement of supports, and pre.- or post-tensioning, internal or external tendons, and cable stays. The time-dependent parameters become central issues in segmentally built bridge construction, since, in many instances, relatively young concrete is subjected to high stresses and thereby significant time dependent deformation.

In the structures capable of analysis by ADAPT the cross section of the bridge can vary along its length. Changes in elevation of the bridge in the vertical plane can be accommodated, but horizontal curves (in plan) of the centroid of the bridge section are not permitted. This is not a practical limitation in most cases, since segmentally constructed bridges are typically straight in plan.

The software has no notable practical limitation on the layout, number and stressing sequence of longitudinal prestressing tendons.

1.3 DISCLAIMER

Considerable time, effort and expense have gone into the development and documentation of ADAPT-ABI. Bridge design in general, and segmentally constructed bridges with time- dependent effects are complex structural engineering tasks. The software should be used by engineers with a good understanding of concrete behavior and structural mechanics. The user accepts and understands that no warranty is expressed or implied by the developers or the distributors in the accuracy or the reliability of the program. The user must clearly understand the basic assumptions of the program and must independently verify the results produced by the software.

1.4 PRINCIPAL STEPS IN DESIGN/ANALYSIS OF SEGMENTAL BRIDGES AND FRAMES

The common steps in the analysis and design of segmentally constructed bridges during the construction phase and when complete are:

1.4.1 Construction Phase

The objective during the construction phase is:

- A. To carry self-weight and construction loads with stresses in concrete limited to permissible construction phase values.
- B. To sustain a code prescribed factor of safety against overload through adequate strength reserve.
- C. (i) To predict the deflection of the structure during each phase of construction, and (ii) to

adjust the formwork for construction of each new segment, such as to control the projected profile of the bridge. The necessary adjustments to the vertical alignment of the frame during the construction is referred to as “camber or adjustment.”

The stress and strength calculations are in many cases straightforward, since the frame under construction may be statically determinate. Free cantilever bridges are good examples of this category. Prior to attachment of the cantilever tips approaching one another from opposite piers, the structure is determinate.

The deflection and camber computations are not always straightforward. A time-dependent analysis is necessary to obtain a realistic estimate of deflections, based on which the camber is determined. The prediction of deflections and the required camber during the construction phase are among strong features of ADAPT-ABI.

1.4.2 Completed Structure

The completed structure may differ significantly in its structural system from the construction phase configuration which may have been statically determinate. With lapse of time, creep, shrinkage and losses in prestressing result in a change in the self-weight and prestressing moments in the bridges. The change in moment with time is referred to as “redistribution of moments,” and is illustrated in **Fig 1.4.2-1** for a free cantilever bridge. The response to live loading is not significantly affected by the long-term effects. The structural design objectives of the completed structure are:

- A. To meet stress limits under service loading (service condition).
- B. To provide adequate strength for safety under design loads (strength condition).
- C. To control the immediate and long-term deflections (typically 20 years).

In summary ADAPT-ABI can be used to obtain the actions (moments, shears, and axial loadings) of the statically determinate and indeterminate pre- and/or post tensioned bridges and frames accounting for the time-dependent effects and displacements. The sectional actions (moment, shear, axial force) obtained from ADAPT-ABI are used to check the stresses of at each cross-section along the bridge. The actions are factored and combined to assess the factor of safety of each section for strength requirements. Post-processor modules of ADAPT-ABI (featured in separate manuals) offer the capability of combinations and enveloping of actions for code stipulated serviceability and strength check.

In all cases the immediate and long-term deflections of the structure modeled are obtained directly from ADAPT-ABI runs.

Chapter 2 of this manual provides a detailed narrative of the software features covered by ADAPT-ABI “Basic.” ADAPT-ABI basic is the core module of the ADAPT-ABI suite of programs. Other modules are essentially post-processors which expand the capabilities of the “Basic”, or present the analysis in user desired formats.

Chapter 3 details the preparation of data and its execution.

Chapter 4 presents the theoretical background to the computations of the program.

Numerical examples are given in Chapter 5.

Specific features of the program, such as details of implementation of time dependant effects are given in the Appendices.

A separate volume, *Example Manual* contains numerous cases in application of ADAPT-ABI.

This Page Left Intentionally

BLANK

LIST OF CONTENTS

2.1	OVERVIEW	1
2.1.1	STRUCTURAL MODEL	1
2.1.2	PRINCIPAL ASSUMPTIONS	4
2.1.3	OUTPUT	6
2.1.4	SOLUTION	6
	A. Node operations	6
	B. Initialization of new frame elements	7
	C. Solution scheme	7
2.2	GEOMETRY	9
2.2.1	OVERALL FRAME GEOMETRY	9
2.2.2	CHANGE IN SECTION ALONG FRAME LINE	9
2.2.3	FLEXIBILITY IN SELECTION OF CROSS-SECTION GEOMETRY	9
2.2.4	BOUNDARY CONDITIONS	10
2.3	MATERIAL PROPERTIES	14
2.3.1	CONCRETE	14
2.3.2	NONPRESTRESSED STEEL (PASSIVE REINFORCEMENT)	14
2.3.3	PRESTRESSING STEEL	15
2.4	PRESTRESSING	15
2.4.1	TENDON NUMBER AND STRESSING	16
2.4.2	TENDON GEOMETRY	16
2.4.3	EXTERNAL TENDONS; UNBONDED TENDONS	16
2.5	LOADING	18
2.5.1	EXTERNAL FORCES	18
2.5.2	EXTERNAL DISPLACEMENTS	18
2.5.3	TEMPERATURE CHANGES	18
2.6	TRAVELER AND FORMWORK OPERATIONS	19
2.7	CONSTRUCTION PHASE	21

This Page Left Intentionally

BLANK

2.1 OVERVIEW

2.1.1 Structural Model

The bridge is modeled as a plane frame consisting of nodes in the global X-Y plane. The nodes may be connected by frame elements (**Fig. 2.1-1**), and springs. In addition by specifying constraints, the free displacement of one node with respect to another can be restricted.

Global displacement degrees of freedom of the nodes consist of X- and Y-translations and Z-axis rotations.

The boundary condition at each nodal degree of freedom may be specified as unrestrained or rigidly restrained, and may be changed at any solution step.

The frame of the structure is generally modeled with concrete *frame elements* (**Fig. 2.1-2**). Each concrete frame element consists of time dependent concrete and a uniformly distributed elastic mild steel component acting in parallel to frame element. Concrete and the uniformly distributed steel model the composite cross section. The mild steel component of the element is intended to model the nominal steel reinforcement usually provided in the elements of a bridge girders and post-tensioned frames. It is not intended to model significant localized reinforcing steel which may be provided to resist global bending moments. Significant localized steel and pre- or post-tensioning can be modeled separately within each concrete frame element.

Each element may be specified to have a uniformly varying cross-sectional area between its ends. However, in the computation, the program converts the stiffness of the specified non-uniform cross-section to that of an equivalent prismatic member of constant cross-section. Short frame element modeling is recommended for increased accuracy for members with a large rate of change in cross-section.

Dead load is automatically computed from the volumetric geometry of the frame element and its applied as equivalent concentrated forces at the nodes.

Each frame element may be installed into and subsequently removed from the structure at any solution step.

Post-tensioned prestressing tendons may be of arbitrary geometry in the X-Y plane. Several tendons may be specified for a frame element. Each tendon is approximated by a number of short, piecewise linear tendon segments defined by tendon points (**Fig. 2.1-1(c)**). Each tendon point is associated with a frame element node, to which its displacements are rigidly constrained. Tendon point global coordinates may be input directly or may be generated automatically using a parametric generation scheme. Initial tendon segment force computation includes the effects of instantaneous prestressing force losses due to friction and anchorage slip (seating loss). Long term changes in tendon force are computed from the tendon's material property together with displacements of the nodes to which tendon is coupled.

It is emphasized that in all cases the two points of each tendon segment must be associated with the end nodes of a given frame element **Fig. 2.1-3**. The program can handle pre-tensioning, post-tensioning, cable stays, and external post-tensioning.

For example in **Fig. 2.1-3(b)** segment two of tendon is associated wrongly with two elements. Diagram 2.1-3(a) shows the correct specification.

Spring elements can be specified with stiffness along their lengths, stiffness perpendicular to their direction and rotational stiffness at their connection to the structure.

Traveling formwork used for casting new frame elements or other construction frames (**Fig. 2.1-4**) are modeled with special frame elements which may be moved around the structure and which are not subject to time dependent strains.

Loading possibilities consist of:

- (i) internally computed frame element dead loads,
- (ii) concentrated actions (forces and moments) acting on the nodes,
- (iii) externally applied displacements, at the nodes (pressure loads)
- (iv) uniformly distributed global X- and Y- direction loads on the frame elements, and
- (v) linearly varying temperature changes through the depths of a frame element, and
- (vi) Linearly varying shrinkage through the depths of an element.

The program has the ability to monitor the actions and displacements of a frame during construction.

All nodes, frame elements, prestressing tendons, springs, stays, and travelers which will ever exist in the structure are first defined to the program prior to the analysis. The construction steps are then described as part of input data. For each construction step the associated elements, plus additional travelers, are tagged for installation or removal. External loading specific to each construction step is described. At the beginning of the analysis, no elements are considered to have been *installed*. Only when a node, element or tendon is *installed* at a construction step it is included in the analysis for that and the following time steps. During subsequent construction steps, elements, tendons, and supports previously installed can be removed, provided the remaining structure is statically stable. When a tendon or element is *undefined* it is not included in the analysis.

An element is *installed* if it has been specifically built into the structure at a construction stage through a BUILD command and has not yet been removed, otherwise the element is *undefined*. A node is defined if at least one defined frame element, spring, or traveler element is connected to it, otherwise the node is *undefined*.

For the analysis of either immediate response (moving live load) or long term effects of completed frames, it is not necessary to go through the construction steps. For completed structures the entire frame is installed in one step and analyzed accordingly.

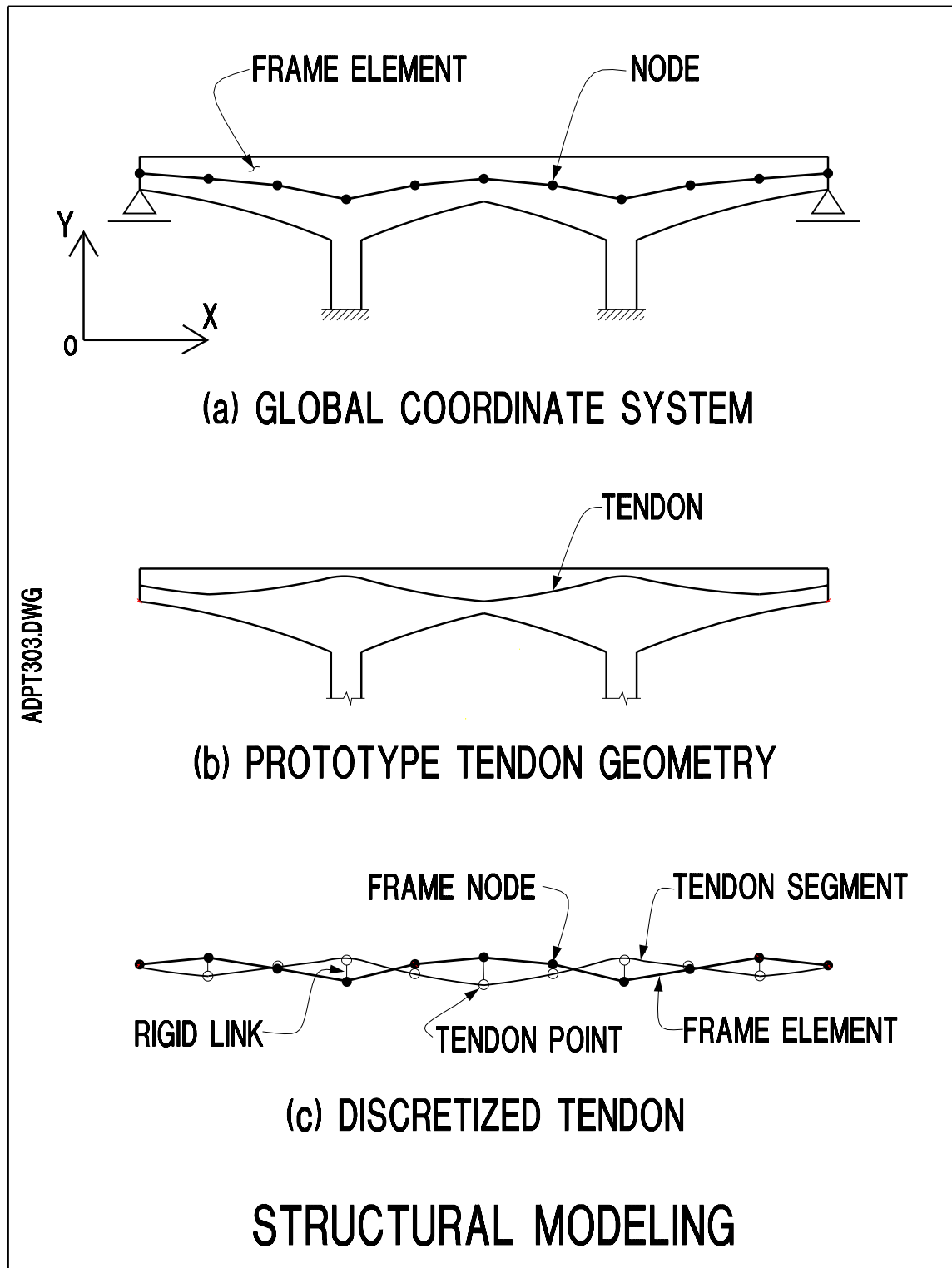


FIGURE 2.1-1

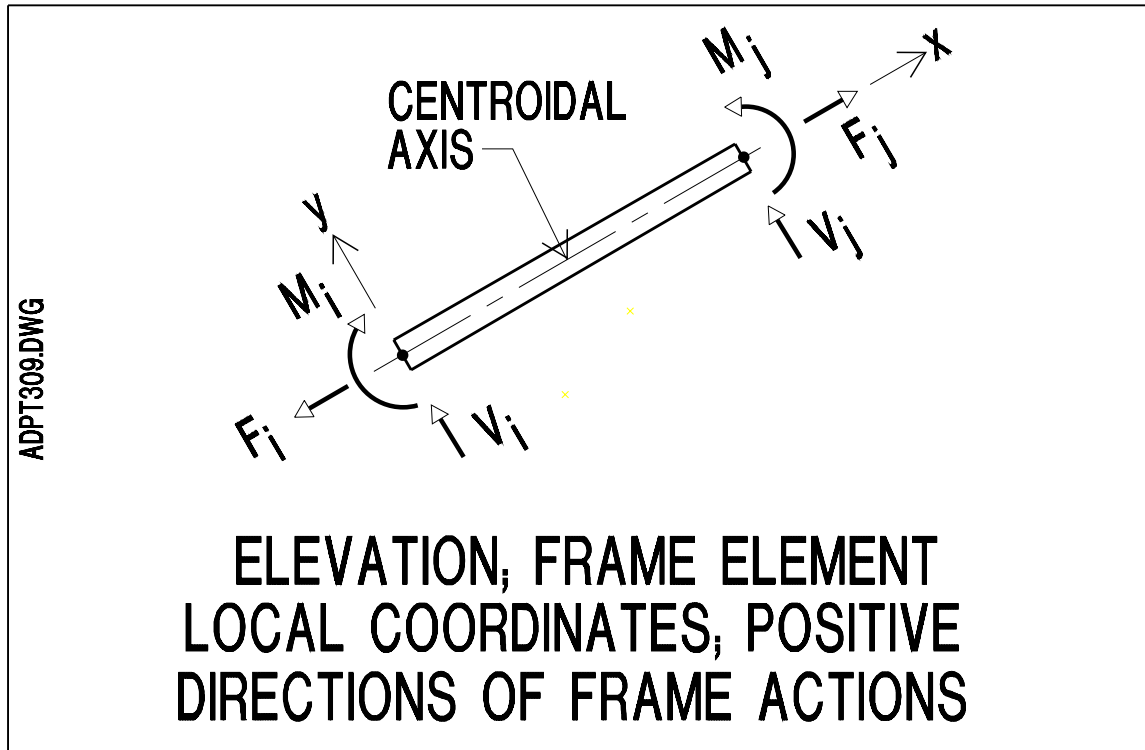


FIGURE 2.1-2

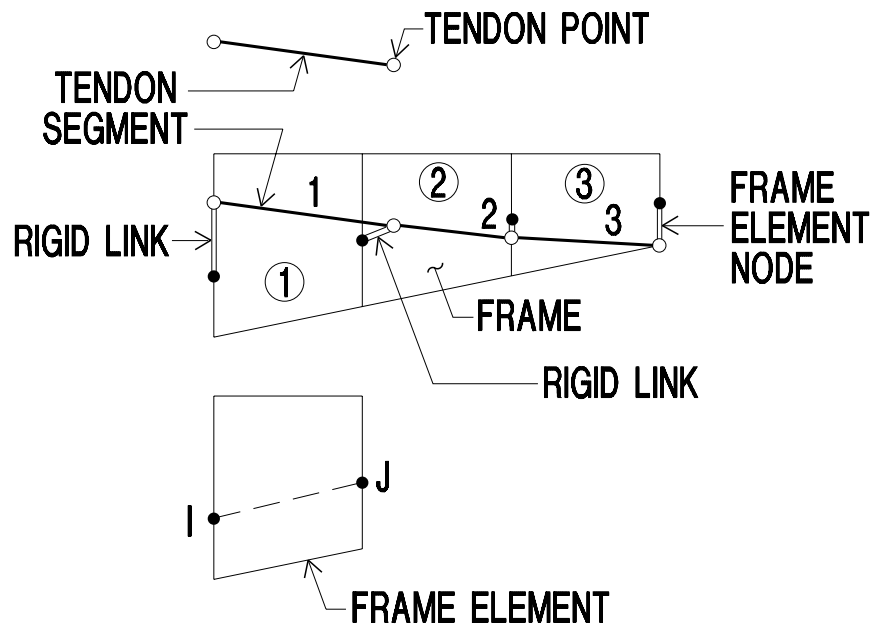
Module ABI-Gen of the programs allows specification of loading at any point on the structure. Further, it has the capability to obtain patterned solutions (skipping of live loading), in order to obtain the maximum value of an action at any points together with its associated other actions, such as maximum moment with associated shear and axial loading.

2.1.2 Principal Assumptions

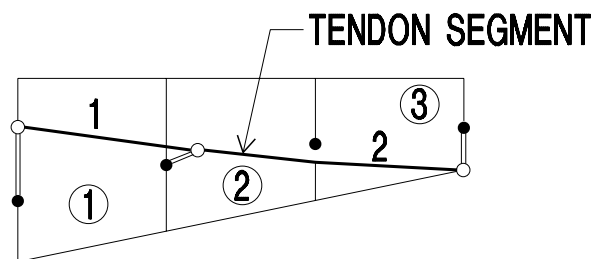
The frame of the structure is in one plane. As a result, all sections are implicitly assumed to be symmetrical with respect to the plane of the frame; all loads are assumed to be applied in the plane of the frame. Torsional effects, if any, do not enter the computations.

The analysis is based on an elastic solution. A solution can be nonlinear because of the time component and the associated nonlinearity in time-dependent material properties. For instantaneous responses, the solution is linear.

The element formulation used is based on classical Bernoulli-Euler beam kinematics; that is to say, plane sections remain plane. The formulation accounts for the effects of axial and bending deformations. Shear deformation is not significant in the class of concrete structures addressed by the program. Hence, shear deformation is neglected. The element formulation for the frame and all other elements used in the program guarantee static equilibrium of total internal forces with the total externally applied loads.



(a) TENDON POINTS AND FRAME ELEMENT NODES



(b) WRONG SPECIFICATION

ASSOCIATION OF TENDON POINTS WITH ELEMENT NODES

FIGURE 2.1-3

Time dependent concrete strains including aging strain, creep strain, shrinkage strain, and temperature strain are automatically considered. Shrinkage and temperature strains may vary linearly over the depth of each element.

As an alternative to discrete specification the mild steel component is assumed to be uniformly distributed over the entire cross section. The frame element consists of parallel concrete and mild steel components, for modeling the typical composite concrete and steel.

Concrete and steel instantaneous stresses are limited to linear elasticity. Since cracking of concrete is not accounted for, solutions which exceed the tensile limit of concrete involve some approximation in deformation.

2.1.3 Output

The output of the analysis consists of:

- A. Mirror image of input data.
- B. Geometry and material properties generated by the program.
- C. Nodal actions (moments, shears and axial loading).
- D. Nodal displacements (translation and rotation).
- E. Reactions at supports.
- F. Stresses in the elements.
- G. Forces in pre-stressing and prestress losses
- H. Prestressing moments, primary moments, and hyperstatic (secondary).
- I. Camber

The volume and detail of printout is controlled by the user when generating input data.

2.1.4 Solution

A. Node operations

Nodal degrees of freedom are included in the equilibrium equations only when the node is attached to at least one currently installed frame element, spring, or traveler element. When no frame, spring, or traveler elements are attached to a node, its degrees of freedom are neglected.

Loads and displacements may be applied only to nodes of the currently installed frame elements. Loads which are applied between the nodes are covered by the program to their equivalents at the nodes. This is an important consideration when removing elements from the structure. Any load remaining on a node which is eliminated from the assembly by removal of all frame and traveler elements attached to it cannot be resisted by the structure and must be removed from the structure before removal of the elements.

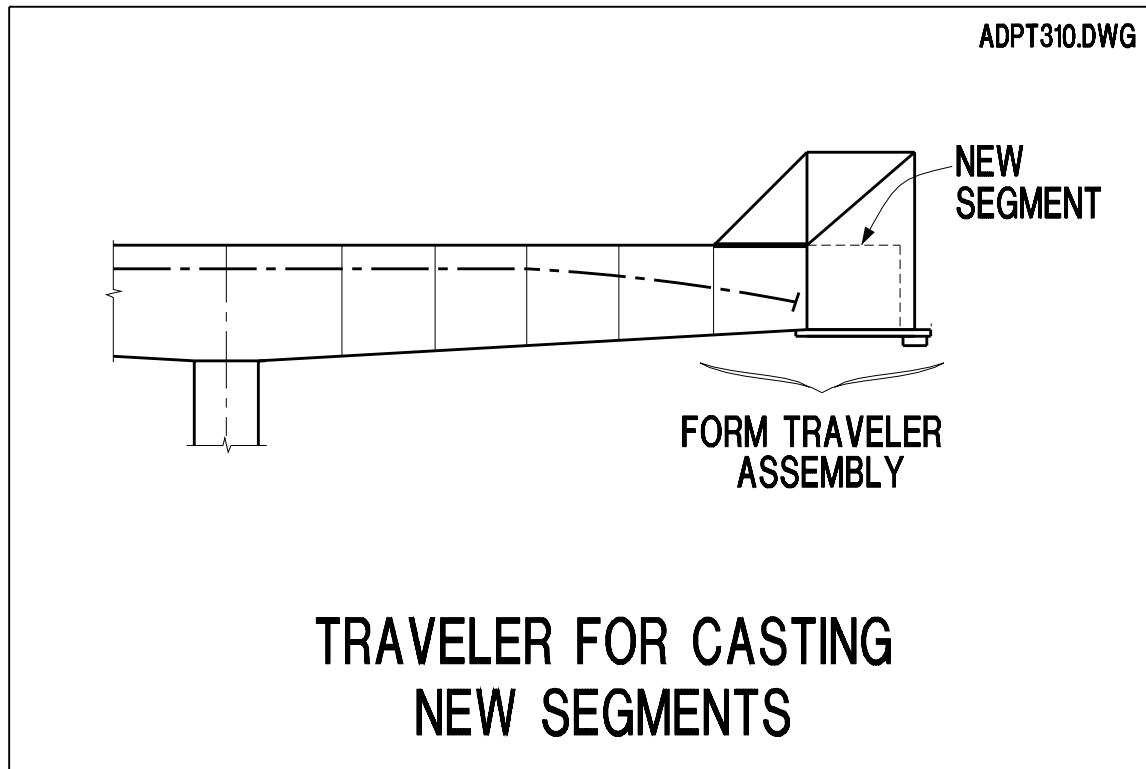


FIGURE 2.1-4

B. Initialization of new frame elements

For regular linear solutions which are commonly used for completed structures, node coordinates entered in input data define the geometry of the frame.

For incrementally constructed frames the addition of new elements accounts for the existing deformation of previously loaded and deformed elements. When an element is installed, in order to provide a statically feasible system one end of the element must be attached either to a restrained node or to a node already connected to another frame element. In most cases this node will have been previously displaced due to previously applied loadings. If the other end of the frame element is attached to a node which is not already connected to another frame element, then the displacements of that node must be initialized based on the existing displacements at the first node. These displacements are automatically initialized by the program, based on the assumption that the new element is a rigid body attached to a previously displaced node (**Fig. 2.1-5**). The initial displacements of the node at the far end is determined from rigid body orientation of the newly installed element.

C. Solution scheme

The solution is based on combining a finite element analysis of the structure with a step forward integration scheme in the time domain. The time domain is subdi-

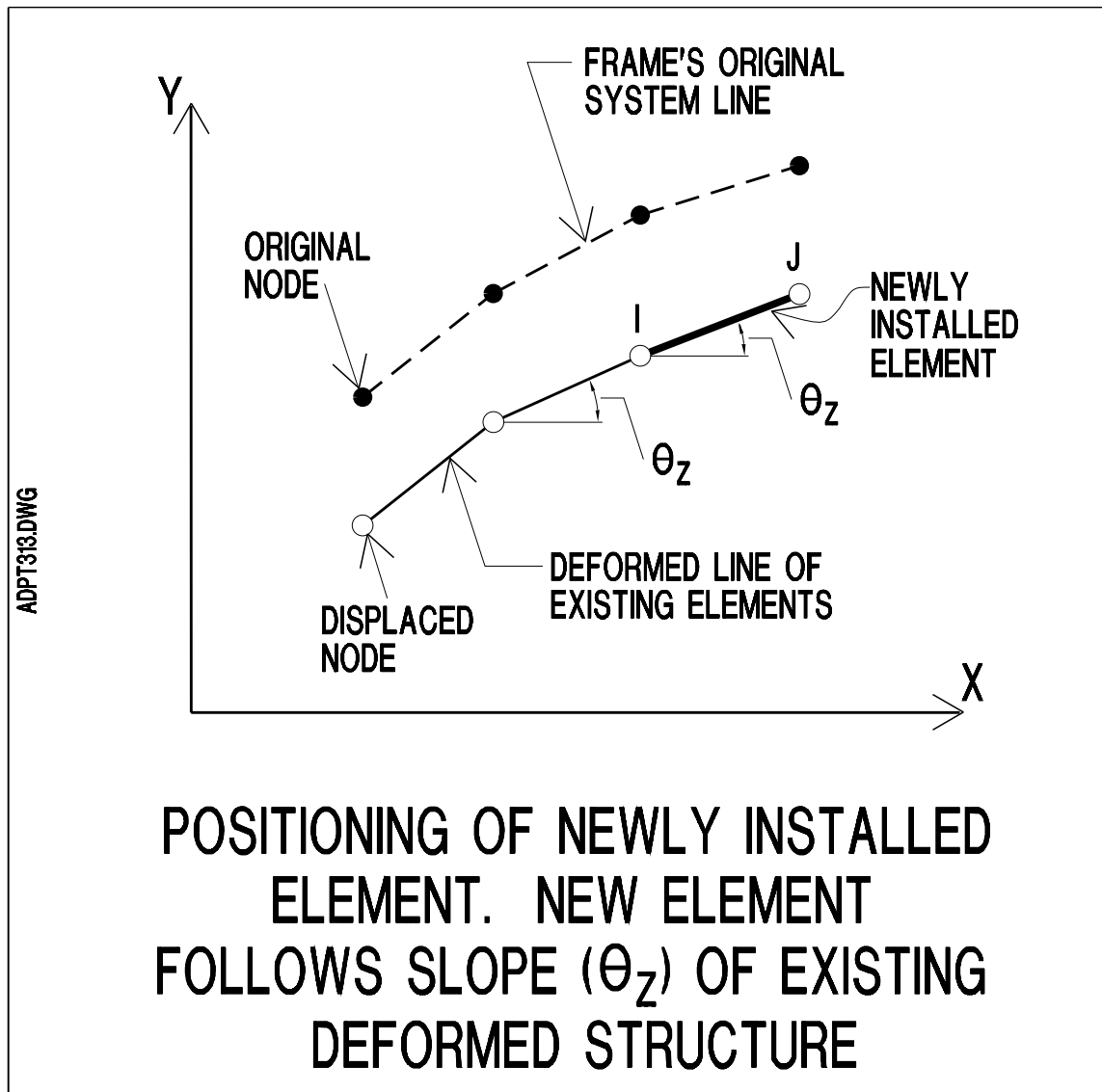


FIGURE 2.1-5

vided into a number of time steps, and an analysis of the finite element system is performed for each step.

Time dependent strains over the time step are considered as an initial strain loading on the finite element system. Time steps may be of arbitrary positive length.

At the beginning of each time step, the complete stress, strain and displacement distribution within the structure is known. Over the length of the time step, any external load increment is gradually applied, and all resulting displacements, stresses and strains in the structure are assumed to vary linearly from their initial values to their final values which are computed by the program. Dead load is automatically included as an external nodal load, unless specifically excluded by user. The linear variation of loading increment and structural response require the use of a zero length time step when instantaneously applied loadings or changes in

the structure configuration are considered. At the end of the specified time step a new stress, strain and displacement distribution within the structure is obtained.

In each solution step, the global equilibrium equations are summed from element components using the direct stiffness method and the principle of virtual work.

Because of nonlinearity of the time-dependent solutions, an iterative procedure is adopted in the solution. The convergence of the solution is fast.

2.2 GEOMETRY

2.2.1 Overall Frame Geometry

The program has no inherent limitation on the frame geometry, so long as all members are in the same plane. Multistory frames and irregular construction with or without prestressing can be treated readily by the software. However, most bridges are multi-span one-level construction, toward which the examples presented herein are primarily focused.

2.2.2 Change in Section Along Frame Line

Both gradual and abrupt changes in cross-section along the frame line can be satisfactorily modeled. The natural frame element nodes are assumed to be at the centroid of cross-section. A gradual change along the frame line, such as illustrated in **Fig. 2.2-1(a)** will result in different coordinates for successive nodes. For each element, the cross-sectional properties at its beginning and end are input. If the change is steep, for increased accuracy, shorter elements should be selected. Also, see the ABI example manual.

If the change is abrupt, such as for a step at the bottom or top of a section (**Fig. 2.2-1(b)**). The nodes representing the centroids of the thicker and thinner elements do not coincide at the interface. To complete the modeling two options are available. If a prestressing tendon passes through the interface of the elements the *node offset* feature of the program is recommended. Otherwise, the *master and slave* feature of the program can be used. In either case, the function of the feature is to couple the otherwise free displacements of the two interface nodes such as to uphold the *plane sections remain plane* assumption.

2.2.3 Flexibility in Selection of Cross-Section Geometry

The local coordinates of the cross-section are shown in **Fig. 2.2-2(a)**. Simple cross-sections can be specified in two ways.

- A. Specify the area A , the moment of inertia I , and the location of the centroid with respect to the top and bottom fibers.

- B. Use the I-section module of the program (**Fig. 2.2-2(b)**) to construct I- rectangular-, T- , or inverted T- sections.

For composite sections, such as a precast section with a cast-in-place topping (**Fig. 2.2-3(a)**), the section is modeled with two parallel elements, each at centroid of one of the section constituents. Observe in part , (a), of the figure and note that the frame elements nodes, the parallel frame elements forming the section are tied by offset constraints. Part , (b), of the figure illustrates the modeling of a three component section, where two offset constraints are introduced at each node.

Application of offset is general and efficient, since it does not result in definition of new nodes.

2.2.4 Boundary Conditions

The boundary condition for any degree of freedom at any node may be specified as either *unrestrained*, *restrained* or *restrained to zero* total displacement. That is to say, the restraint can be imposed throughout the entire construction phase (zero total displacement), or it can be imposed starting at a defined phase in construction.

Each node in the frame is initially assigned three degrees of freedom. An *unrestrained* (free to displace) degree of freedom is assigned a corresponding equation number in the global equilibrium equations, and its displacement increment in subsequent solution steps is computed by the program. Nodal loads may be applied to unrestrained degrees of freedom, and the program determines the response of the structure to these loads. Externally applied nodal displacements may not be applied to unrestrained degrees of freedom.

A *restrained* (rigidly supported) degree of freedom has no corresponding equation in the global equilibrium equations, and its displacement increment in subsequent solution steps is set to zero. Reactions at restrained degrees of freedom are summed by the program from external nodal load and element internal resisting force components. If the boundary condition is subsequently changed to unrestrained, the reactions are automatically applied as part of the loading increment on the structure. Nodal loads may be applied to a restrained degree of freedom, but these loads are resisted by the reactions and do not influence the stresses in the structure. Externally applied nodal displacements may be applied to any restrained degrees of freedom.

A *restrained to zero* degree of freedom is a special case of the restrained case discussed above. The only difference between the two is that in the restrained to zero case, an external displacement is applied in the first solution step after the specification is made, so that the total displacement in the degree of freedom is zero.

Rotational and extensional *spring supports* are simulated through spring elements.

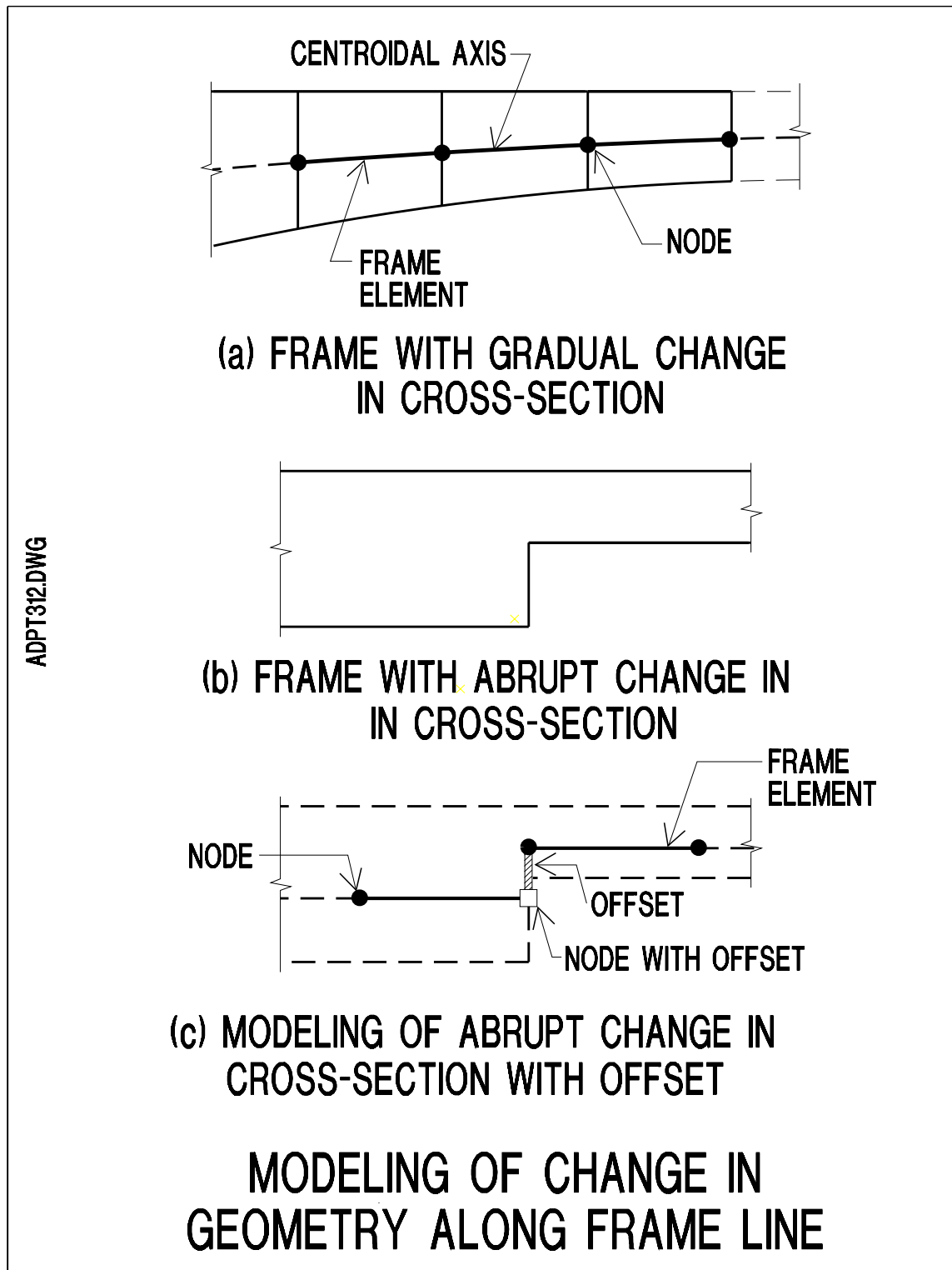
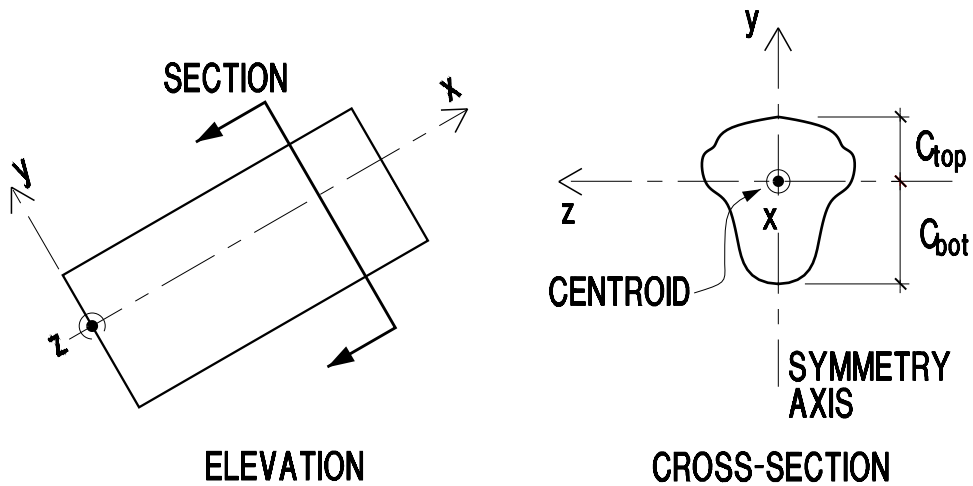
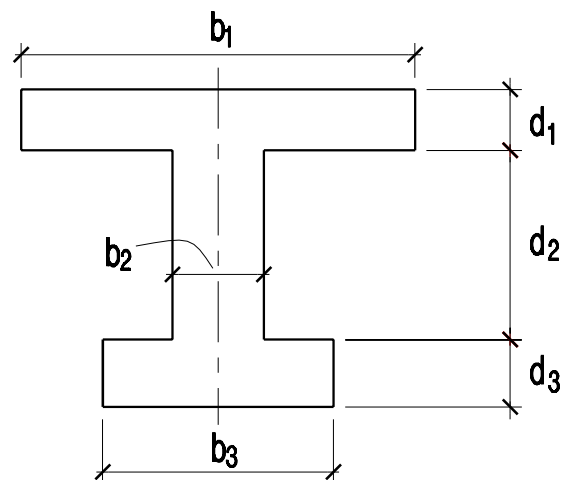


FIGURE 2.2-1



(a) ELEMENT ELEVATION AND SECTION



(b) BASIC LIBRARY CROSS-SECTION

ELEMENT COORDINATES AND THE BASIC CROSS-SECTION MODULE

FIGURE 2.2-2

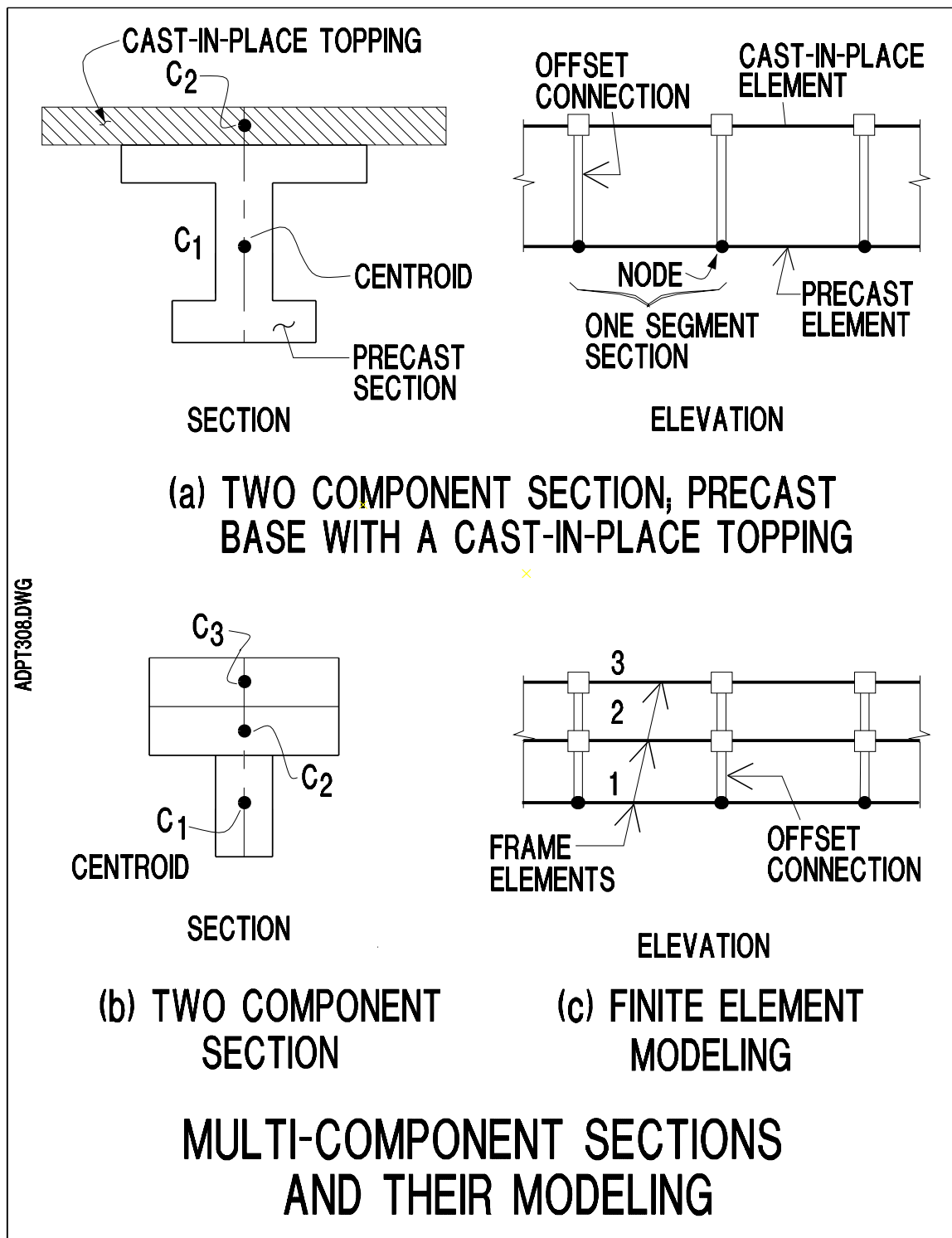


FIGURE 2.2-3

2.3 MATERIAL PROPERTIES

The materials to be defined are *concrete*, *nonprestressed steel*, and *prestressing steel*. Generally several different types for each material can be defined.

The multi-layer modeling of a cross-section is of particular value when the effects of nonlinear temperature gradient through a section is sought. Depending on the accuracy required, the section can be modeled as several layers of identical material. For each layer, the temperature gradient is linear.

Another typical application is in composite construction, where a portion of the section is cast at a later stage.

2.3.1 Concrete

The program features comprehensive formulation in the definition of concrete material properties and their implementation in the computation of concrete-related time-dependent displacements and actions. A close estimate of deformations becomes critical on the impact of time and construction schedule on the servicability and strength of the structure.

For each concrete material used, two groups of information are defined:

A. *Creep and Shrinkage Properties.*

The Concrete Creep and Shrinkage Parameters define the mathematical model to be used for the computation of the long-term effect due to creep and shrinkage. The material models included in the software are based on:

- (i) ACI
- (ii) CEB/FIP
- (iii) Laboratory obtained, or user defined according to other concrete codes

B. *Concrete Properties.*

These include parameters defining the specific concrete properties of each concrete material used. These are: crushing strength; weight; coefficient of thermal .

Aging of concrete is calculated using the parameters given for concrete properties.

2.3.2 Nonprestressed Steel (Passive Reinforcement)

Nonprestressed steel is identified by its modulus of elasticity, thermal coefficient of expansion, and its percentage in the cross-section. Each frame element can be assigned its own type and percentage of steel. Nonprestressed steel is considered as a linear elastic material, uniformly distributed over the entire cross-section of the frame element.

The amount of nonprestressed steel available in any cross-section is expressed by a steel percentage coefficient.

Significant concentration of nonprestressed steel is modeled as steel bars of given cross-section and properties located at user defined locations in a frame element.

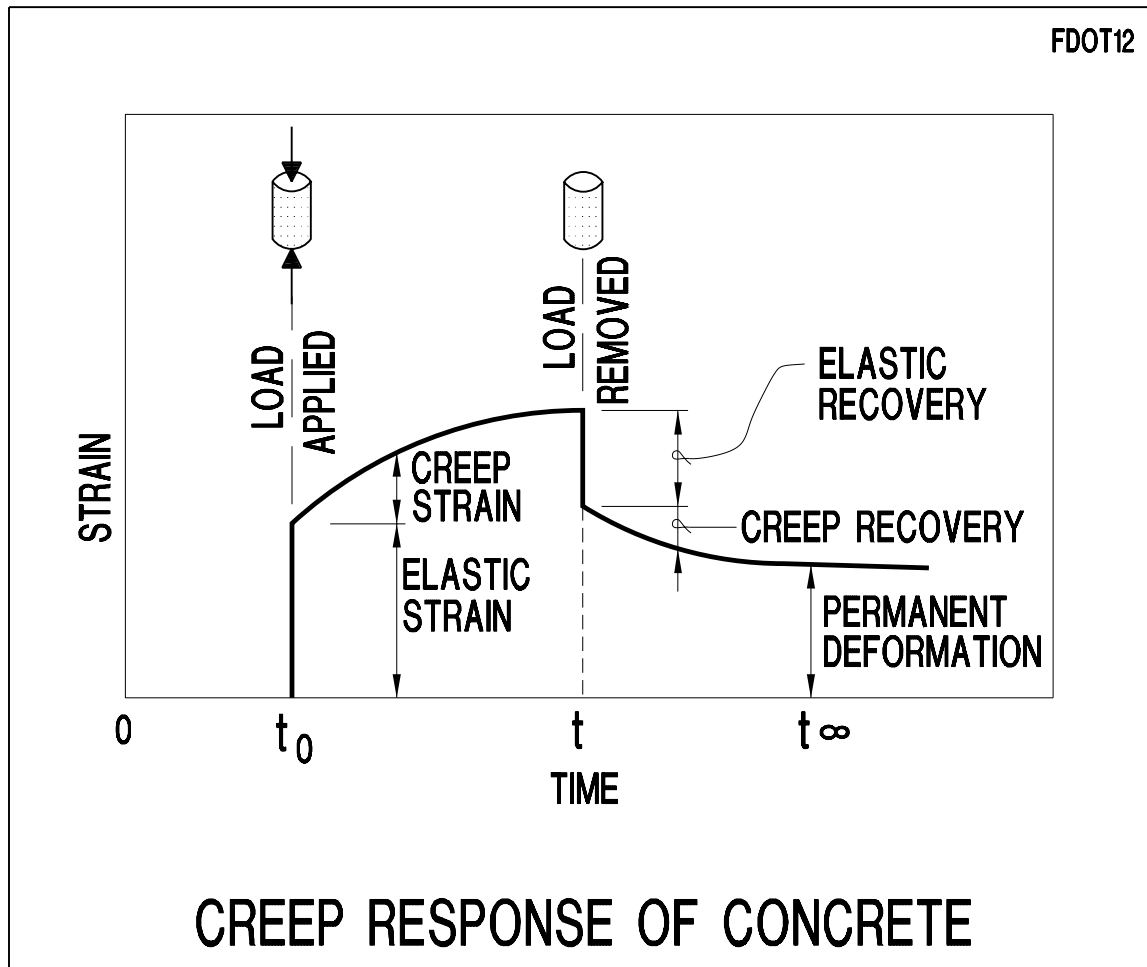


FIGURE 2.3-1

2.3.3 Prestressing Steel

For each prestressing steel, its modulus of elasticity, ultimate strength, yield strength, stress relaxation coefficient and thermal coefficient of expansion are defined. As an alternative laboratory measured relaxation values may be given as input data.

2.4 PRESTRESSING

Prestressing is achieved through definition of tendons and subsequently stressing of these tendons. Tendons can be pretension, bonded, unbonded and stay.

2.4.1 Tendon Number and Stressing

Multiple number of tendons, each with its own geometry, material properties, and force characteristics can be defined. Prestressing tendons defined are activated by *installing* them in the construction sequence. When installed, the tendon's contribution to the stiffness of frame elements is implemented in assembly of the next global equilibrium equations. In all subsequent steps the tendon's current stiffness matrices, relaxation strain equivalent loads and current total forces are included. This simulates a tendon which is unbonded during stressing and perfectly bonded thereafter. For unbonded post-tensioned systems the tendon need be encased in a sheathing as described in section 2.4.3.

Prestressing tendons are removed by eliminating the contributions of their stiffness and internal forces in subsequent global equilibrium equation assembly.

Initial forces in the tendon segments are computed including the effects of friction and anchor set. Each tendon may be stressed from either one or both ends.

Long-term stress losses are effected through tendon's relaxation coefficient and the frame displacements to which the tendon is locked. These include the effects of creep and shrinkage among other causes of strains in a frame element.

2.4.2 Tendon Geometry

The only limitation on a tendon's geometry is that it is modeled as a straight segment between adjacent frame nodes to which it is associated. At installation, the location of a tendon with respect to a node is defined by its offset distance from the node. Throughout the solution the tendon offset distances are retained as constant.

There are a number of predefined library shapes which can be used to describe the geometry of regular tendons. **Fig. 2.4-1** illustrates several of the profiles included in the library of the program.

For an irregular tendon, the geometry is defined by giving the coordinates of the tendon along its path.

2.4.3 External Tendons; Unbonded Tendons

The external and unbonded tendons are modeled with a sheathing which serves to keep the tendon isolated from the frame elements. At locations where a node of the sheathing coincides with that of the frame structure, the external or unbonded tendon is assumed to have been securely connected to the frame. Force and displacement compatibility between the tendon and the frame structure is established at the connection points between the tendon sheathing and the frame structure defined by the user.

The sheathing of external and unbonded tendons is itself modeled as a frame element with negligible stiffness. An example in the Examples Manual illustrates the application of unbonded tendons.

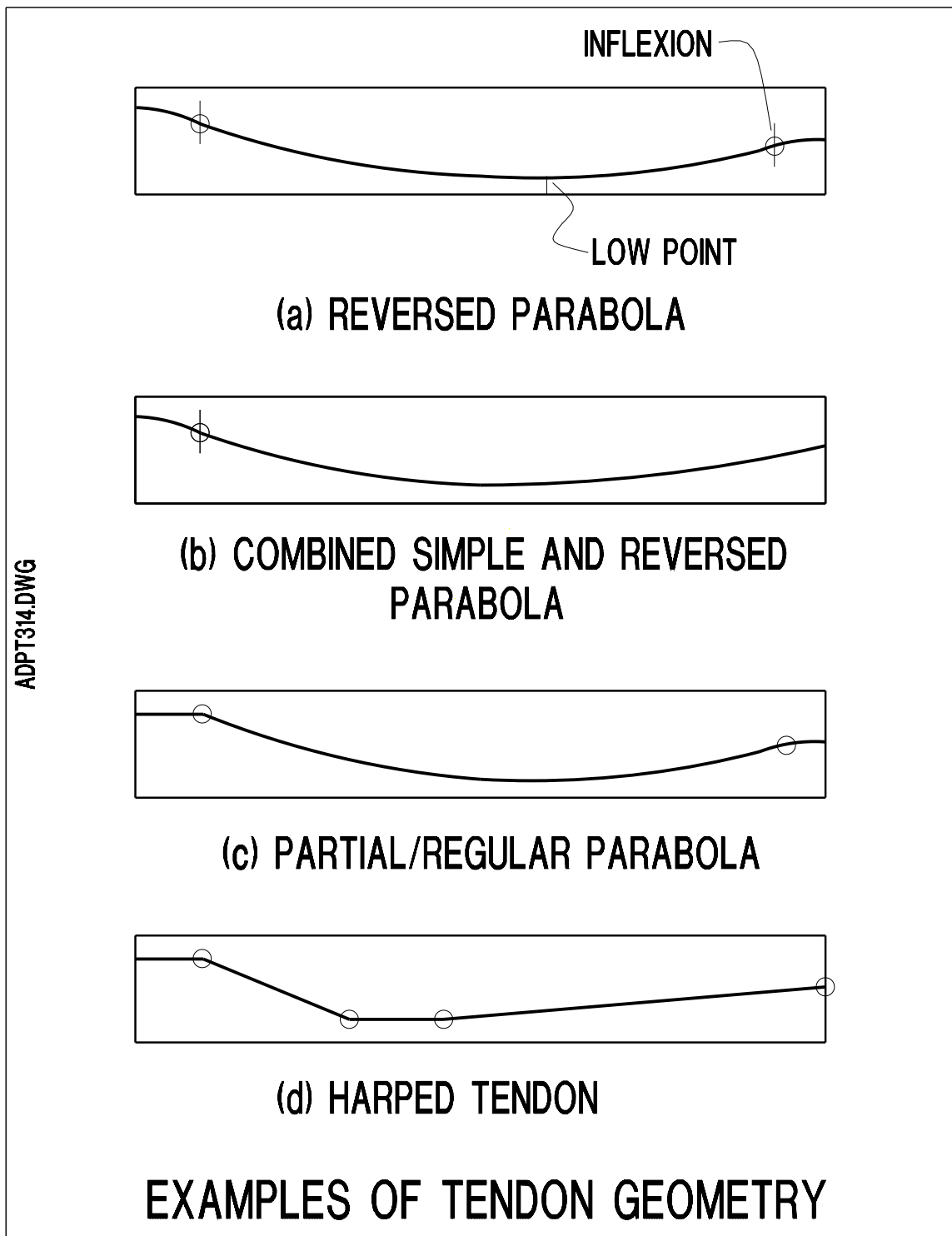


FIGURE 2.4-1

2.5 LOADING

2.5.1 External Forces

Concentrated external forces and moments may be applied on the frame element nodes. Uniformly distributed loads may be applied on the frame elements. These loads can be applied only to currently installed frame elements and the nodes to which they are connected. Concentrated loads specified to act between the nodes (ABI-Gen Module) are automatically converted to nodal loads by the program.

Concentrated nodal loads in the global directions are summed directly into the total external load vector. Uniformly distributed frame element loads in the global directions are first converted by the program to equivalent nodal actions (forces and moments) and are then summed into the total external load vector. These loadings remain on the structure until they are removed by application of an equal but opposite load.

Dead load of frame elements and travelers is automatically applied as a set of equivalent nodal forces. These forces are computed based on the calculated length and cross sectional area of the frame element, and the unit weight of the concrete. Gravity load multipliers, which are factors by which the dead load is multiplied, before summing into the total load vector, may be specified for gravity load in the global X- and Y- directions. Changing the gravity load multipliers changes the dead load factors for all frame elements currently installed in the structure. Thus the entire dead load can be removed from the structure by specifying zero gravity load multipliers.

2.5.2 External Displacements

External displacements and rotations may be applied on restrained degrees of freedom of the nodes. These displacements and rotations can be applied only to nodes connected to currently installed frame elements, and remain in effect until they are removed by application of an equal but opposite displacement or rotation, until all elements connecting to the node are removed or until the restraint is removed from the degree of freedom.

The displacements and rotations are applied to the structure as an initial strain loading case in the elements. Each frame, tendon and traveler element connected to a node with an imposed displacement is considered to be under an initial strain equal to the strain resulting from the imposed displacements acting on the element with all other element degrees of freedom fixed.

This method for considering externally applied displacements and rotations eliminates the numerical problems associated with the use of stiff external springs and applied forces for simulating applied displacements.

2.5.3 Temperature Changes

Temperature changes and gradients may be applied to any currently installed frame element, and remain in effect until they are removed by application of an equal but

opposite temperature change or gradient. When a temperature change or gradient is applied, its specification is by frame element number, but the temperature change is actually applied to all tendon elements passing through the frame element in addition to the frame element itself.

The temperature change is applied to the structure as an initial non-mechanical strain loading in the elements. Each frame and tendon element is considered to be under an initial strain equal to the strain resulting from the temperature change acting on the element with all element degrees of freedom fixed.

The temperature variation within an element is specified by the temperatures at the extreme fibers of the frame element. The internal temperatures are assumed to be constant over the length of the element and to vary linearly over the depth of the element. For the frame element, this temperature field is used directly to compute the initial strains. For prestressing tendon elements, the element is assumed to be under constant strain so the temperature at the depth corresponding to the mid-length of the tendon segment is used for computing the initial strain. The initial stress-free temperature for each element may be specified independently for each element.

If the temperature variation through the depth of the frame structure at any given section is not linear, that frame section must be modeled as a multi-layered section. Combination of linear variations of temperature through each layer model the imposed nonlinear temperature change.

2.6 TRAVELER AND FORMWORK OPERATIONS

Traveler is the generic name used for an auxiliary structure which is employed during the construction for positioning and supporting the weight of new segments, or lifting of new segments into position. A traveler is generally assembled and secured to the constructed portion of the bridge adjacent to the position of new construction. The weight of the traveler and the new piece to be added is reacted by the existing portion of the bridge. A simplified traveler extending over one segment is shown in **Fig. 2.1-3**. Unlike a gantry (**Fig. 2.7-1**) a traveler is commonly short.

Fig. 2.6-1 illustrates the idealization of a traveler which extends over three segments. Part (a), of the figure shows the traveler positioned for receiving the weight of a new segment. Part (b), of the figure shows the traveler elements. Note that in this example the traveler is made up of three elements. When in position, the traveler is locked into the bridge frame elements at nodes I, J and K shown in the figure.

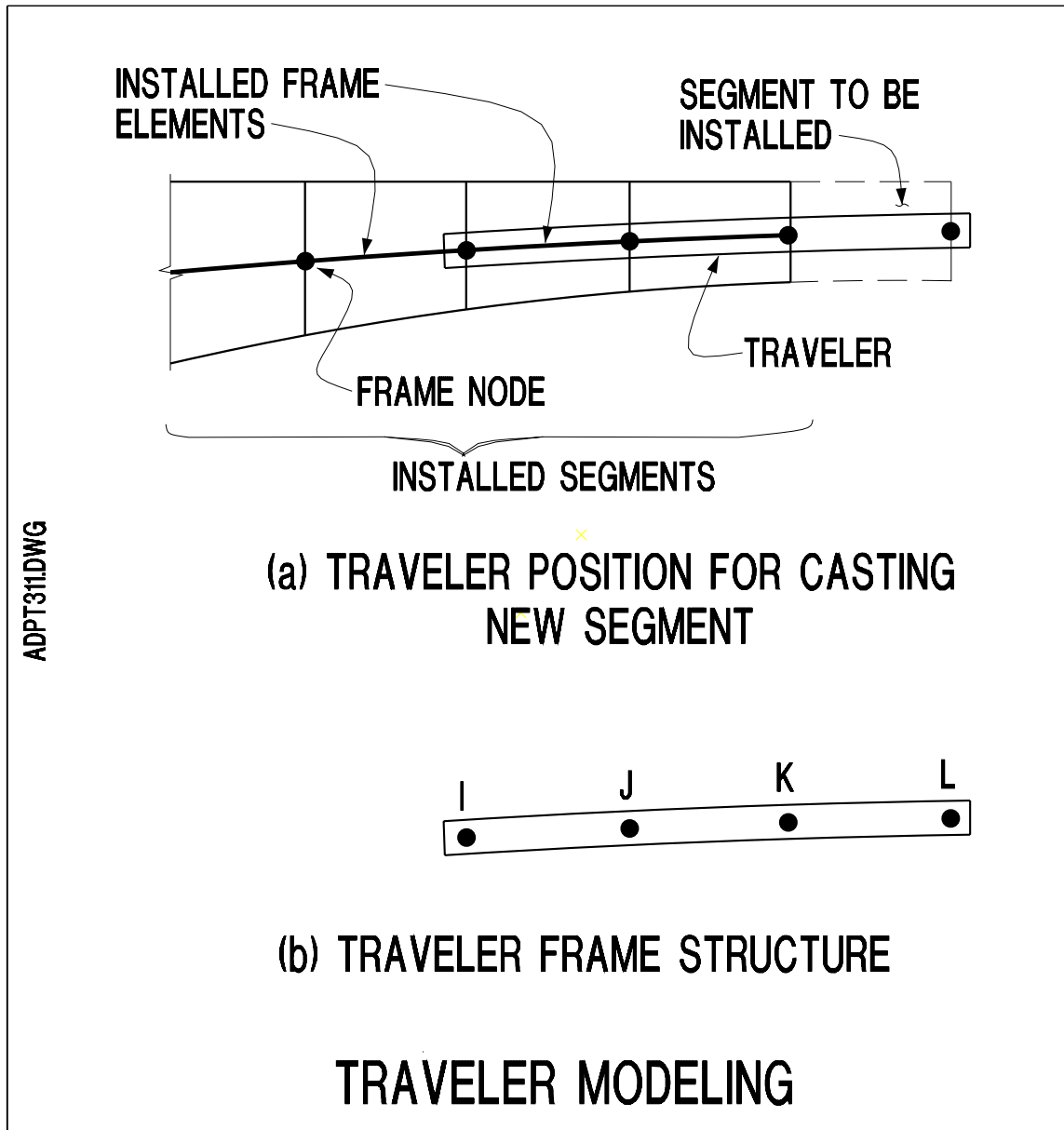


FIGURE 2.6-1

In the solution, travelers are installed at a particular location by including the contributions of their current stiffness matrices, dead loads and internal stresses in subsequent global equilibrium equation assemblies. Travelers are removed by neglecting the contributions of their stiffness, dead load, and internal stresses in subsequent global equilibrium equation assemblies.

Travelers are automatically moved by the program by first removing them from their current location and then installing them in the new location. When a traveler is first moved to a new location, its element characteristic matrices are computed for the element geometry at the new location. Traveler elements are always assumed to be under zero stress when they are installed at a new location. Traveler elements behave elastically and have no *memory* of previous locations.

For any given problem, the total weight of a traveler is assumed to remain constant, regardless of its location in the structure. When a traveler is installed at a new location, its equivalent nodal dead loads are computed based on the total weight of the traveler on a tributary area basis. The final deflection and camber of a structure is generally very sensitive to the stiffness of the traveler. It is essential to include realistic values for traveler parameters in the analysis.

2.7 CONSTRUCTION PHASE

In addition to the analysis capability of completed structure, the program can trace the displacements and stresses during the construction phase. The construction sequence is defined through a number of construction steps. At each step, which is differentiated from a previous step by lapse of time and/or construction and loading operations, a complete solution is obtained.

Over the length of any construction time step, tendon and frame elements may be installed or removed from the structure, traveling formwork may be installed, moved or removed from the structure and nodal boundary conditions may be changed. This permits the analysis of structures built with a variety of segmental erection schemes, including cantilever construction (**Fig. 2.7-1**), span by span construction (**Fig. 2.7-2**), incrementally launched construction, and pre-cast girder cast-in-place deck construction. **Figs. 2.7-3**, and **2.7-4** illustrate examples of other incremental construction techniques in use.

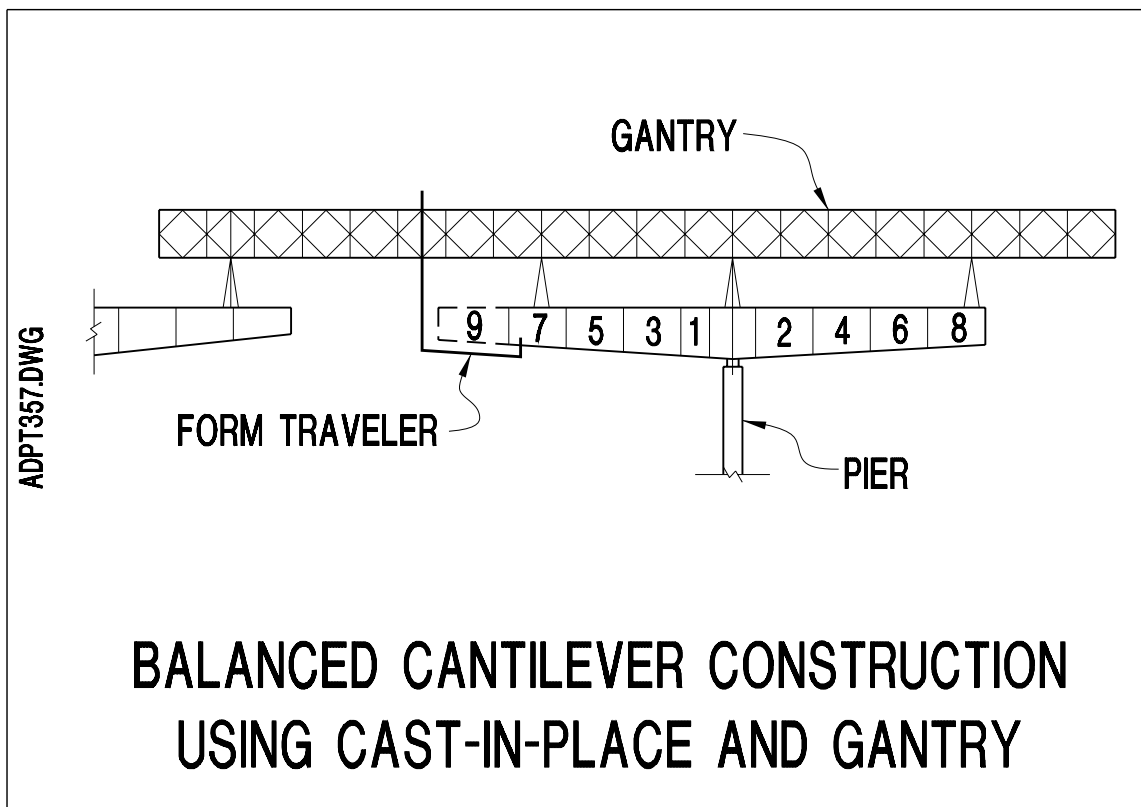


FIGURE 2.7-1

For cantilever and span by span construction schemes, the global X-Y coordinate system can be considered fixed in space. The construction operations described below can be applied directly, and the structure can be built in the computer following the same operations used in the actual construction.

For incrementally launched construction schemes, the global X-Y coordinate system should be considered as a reference system which moves with the girder. Then, to model the launching operations, the supports are moved along the girder by restraining and releasing nodal boundary conditions.

Since construction operations are usually considered as instantaneous and not gradual, a zero length time step is required when a construction operation is performed.

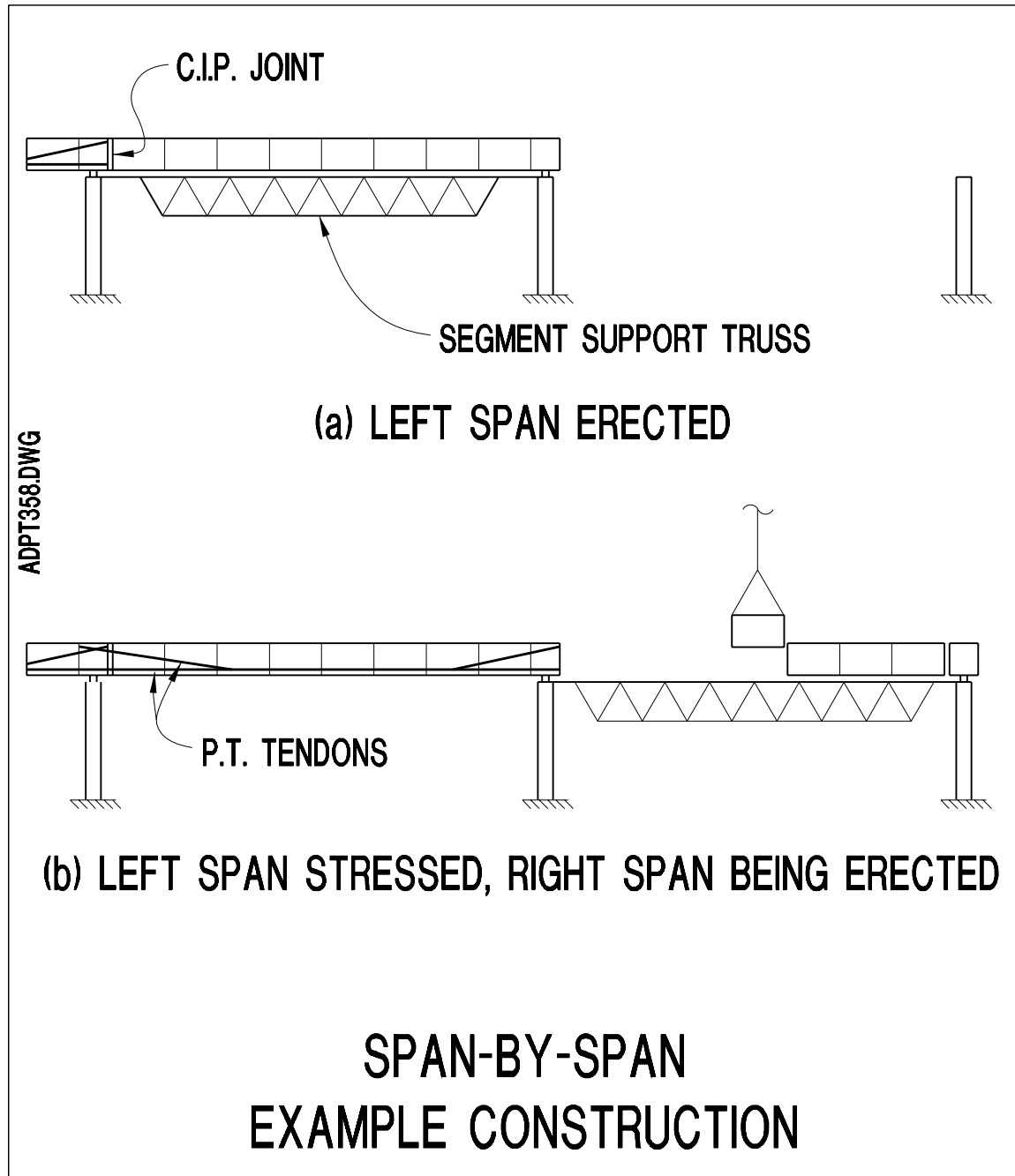


FIGURE 2.7-2

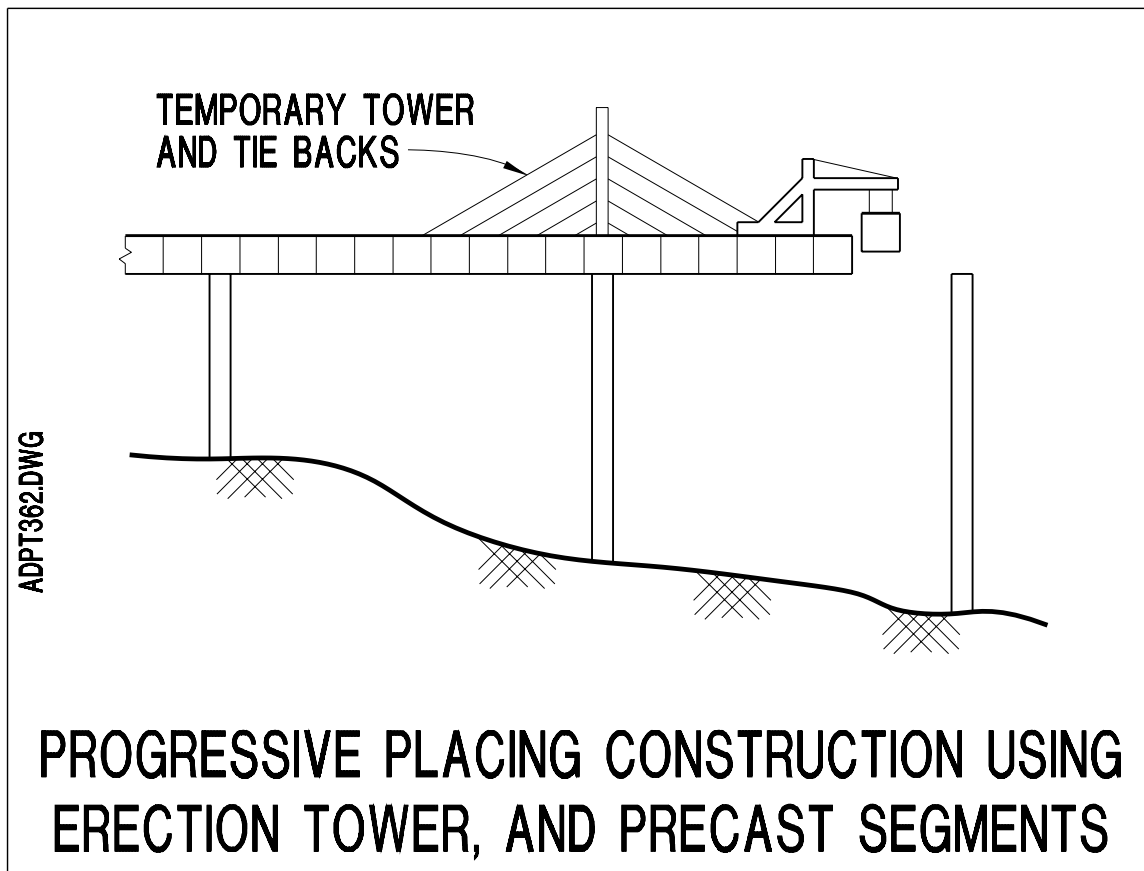


FIGURE 2.7-3

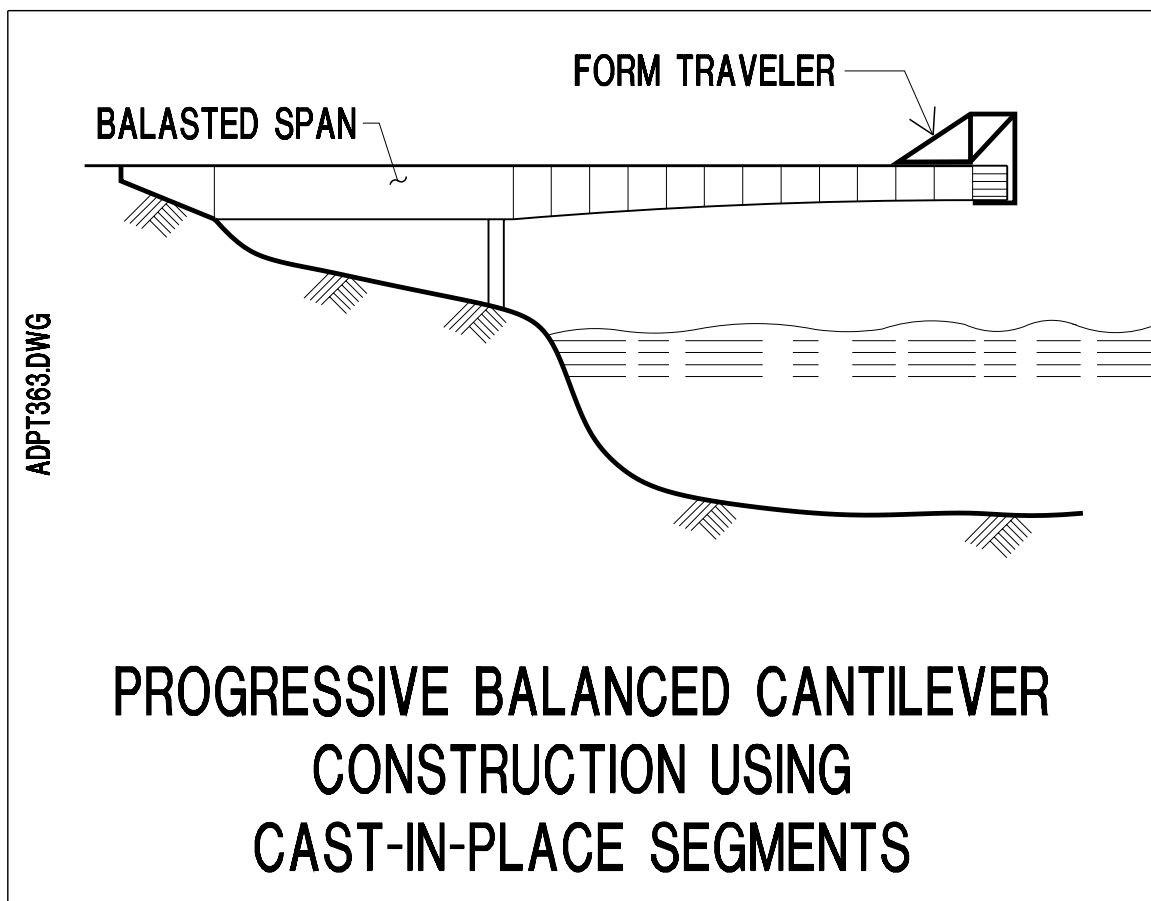


FIGURE 2.7-4

This Page Left Intentionally

BLANK

LIST OF CONTENTS

3.1	OVERVIEW	1
3.1.1	CAPABILITIES	1
3.1.2	PREPARATION OF INPUT DATA	1
3.2	COMMAND SUMMARY	2
3.2.1	NATURAL SEQUENCE OF COMMANDS	2
3.2.2	SUMMARY OF COMMANDS	3
3.3	SAMPLE INPUT FOR A NONPRESTRESSED CANTILEVER BEAM	5
3.3.1	DESCRIPTION OF THE STRUCTURE	5
3.3.2	INPUT FILE	6
3.4	SAMPLE INPUT FOR A PRESTRESSED CONCRETE BOX GIRDER BRIDGE	7
3.4.1	DESCRIPTION OF THE STRUCTURE	7
3.4.2	INPUT FILE	8
3.5	DETAILED DESCRIPTION OF COMMAND SYNTAX	10
	ACTIVATE EXTRACT	10
	BUILD	10
	CAMBER	11
	CHANGE COMPLETE	12
	CHANGE STRUCTURE	12
	CONCRETE PARAMETERS	13
	CONCRETE PROPERTIES	23
	DE-STRESS	25
	ELEMENTS	26
	FRAME	26
	LOADING	31
	MESH COMPLETE	35
	MESH INPUT	35
	MILD STEEL PROPERTIES	35
	MOVE	36
	NODES	37
	OFFSET DATA	38
	OUTPUT	39
	PRESTRESSING STEEL	41
	REMOVE	42
	RESTRAINTS	42
	SECTION PROPERTIES	45
	SEQUENCE	47
	SET	49
	SOLVE	50
	SPRINGS	51
	START	52
	STAY ANALYSIS	53
	STOP	54
	STRESS	54
	TENDON GEOMETRY	56

TITLE	62
TRAVELERS	62
UNITS	62

3.1 OVERVIEW

ADAPT-ABI software is a special purpose structural analysis program, specifically developed for the time-dependent analysis of segmentally erected prestressed concrete plane frame structures and bridges. It accounts for variation with time of creep, shrinkage, relaxation in prestressing and concrete aging, as well as creep recovery. The structure is modeled as a system of frame elements and prestressing tendon elements connected at nodes. Time is divided into a number of time steps, and the program computes the response of the structure at the end of each of these time steps.

3.1.1 Capabilities

The program can *build the structure in the computer* using any statically feasible construction sequence for the plane frame. Frame elements, springs, and tendon elements may be installed and removed at any time step during the solution. Prestressing tendons may be stressed from either one or both ends, and may be subsequently restressed or removed. Traveling formwork may be moved to any location. Nodal loads may be applied or removed at any time step. Nodal boundary conditions may be changed from fixed to free or free to fixed at any time step. All actions are considered to occur linearly (gradually) over the duration of the time step. Thus any instantaneous action must be modeled with a zero length time step.

The solution includes the effects of creep, shrinkage and aging of concrete, plus friction, anchorage slip relaxation of the prestressing steel a change in stress in prestressing steel due to deformation of the structure. The underlying theory and its numerical implementation in the program are discussed in Chapter 4 and the appendices of this manual. Material constitutive parameters may be computed by the program to model the time dependent material properties according to laboratory measurements (user defined) , the ACI, or the CEB/FIP recommendations.

Output can include incremental and total nodal displacements and reactions, frame element stresses and sectional actions, prestressing tendon forces and resultants. Numerous options exist for the specification of the type of output desired during any solution or output phase.

3.1.2 Preparation of Input Data

Input data are provided to the program from an input (ASCII) file. All input takes the form of commands consisting of one or more keywords, followed by optional numerical data required for the execution of the command. The actions and the data requirements of the commands are summarized in the following pages. All keywords must be typed in full as they appear in this manual. Mis-spelled words result in error. In most cases the errors are detected by the software's data verification module.

All numerical data are entered in the following free field form:

n1,n2,n3, . . . ni A=a1,a2,a3, . . . ai B=b1,b2,b3, . . . bi;

where n_i , a_i and b_i represent input data and the character pairs $A=$ and $B=$ are identifiers, specified in the input manual, for the data list which follows. Items in a data list must be separated by a single comma or by one or more blanks. If a numerical data list requires no identification, such as $n_1, n_2, n_3, \dots$ in the above example, the list must be located as the first data list on the line. A data set of the form $B=b_1, b_2, b_3, \dots$ (preceded by a pair of identities, " $B=$ " in this case) may appear in any order or location on the line.

Simple arithmetical statements are possible when entering floating point real numbers. For example, the following forms of data can be entered:

$E=29600*144 \quad C=200*12+3.5, \quad 400/12, \quad 10+20/5-2$

These arithmetical statements are evaluated from left to right without operator hierarchy. The statement $10+20/5-2$ is evaluated as $((10+20)/5)-2$. Blank spaces within arithmetical statements are not allowed.

The command interpreter in the program recognizes several special characters used to delimit multiple command lines on one physical line, continuation of a command line on the next physical line, and comments to be ignored by the command interpreter.

- ! The exclamation mark is a special character used to delimit several command lines provided on one physical line. The data to the right of an exclamation point are not considered to be part of the current command line, and are instead considered as the next command line. Any number of command lines may be input on one physical input line by separating them with this character.
- \ The backslash is a special character used for continuation of a command on the next physical line. All data to the right of the backslash are ignored, and the following input line is interpreted as a continuation of the first line. This option allows a maximum of 160 characters to be entered as one line of data. The maximum, number of characters permitted on any physical line is 80.
- ; The semicolon is a special character used to delimit comments in the input stream. All characters to the right of the semicolon up to the end of the line are ignored by the program. If the semicolon is located in the first column in a command line the entire command line is ignored.

3.2 COMMAND SUMMARY

3.2.1 Natural Sequence of Commands

The following represents the sequence of commands for most common problems. Not all the commands may be necessary in a given problem.

```

START
TITLE
UNITS
ACTIVATE EXTRACT;1
CONCRETE PARAMETERS

MESH INPUT
  NODES
  .....blank line .....
  SEQUENCE
  .....blank line .....
  CONCRETE PROPERTIES
  MILD STEEL PROPERTIES
  SECTION PROPERTIES
  OFFSET DATA
  ELEMENTS
    FRAME
    SPRINGS
  .....blank line .....
  PRESTRESSING STEEL
  TENDON GEOMETRY
  .....blank line .....
  TRAVELERS
MESH COMPLETE

SET

CHANGE STRUCTURE
  BUILD (Frame Element)
  RESTRAINTS
  .....blank line .....
  REMOVE (Element)
  STRESS (Tendon)
  DE-STRESS (Tendon)
  MOVE (Traveler)
CHANGE COMPLETE

LOADING
.....blank line .....
SOLVE
OUTPUT
CAMBER
STOP

```

3.2.2 Summary of Commands

The following brief descriptions of each command and its input data requirements are intended as a summary of the software's command syntax and as a quick reference guide for the experienced user. Detailed descriptions of each command and the meaning of all data are provided further on in this chapter. Sample inputs for a simple cantilever and a simple prestressed concrete bridge are included in this chapter for illustration.

¹ Command necessary if ABI-GEN is used.

COMMAND SUMMARY	DESCRIPTION
Problem Initialization Input:	
START	Start interpreting new input data
START R=1	Start interpreting data for a program re-run
TITLE N=?	Print N lines of title
UNITS U=?	Set input/output units
ACTIVATE EXTRACT	
CONCRETE PARAMETERS N=?	Number of concrete materials used followed by the modeling scheme used for concrete's creep and shrinkage
Structure Definition Input:	
MESH INPUT	Start definition of structural system
NODES	Input node coordinates
SEQUENCE	Optional input for sequence of degrees of freedom for solution optimization
CONCRETE PROPERTIES	Input concrete material properties; strength; time dependent parameters; material model type
MILD STEEL PROPERTIES	Input mild steel properties and its amount in concrete used
SECTION PROPERTIES	Input cross section properties or geometry
OFFSET DATA	Input information on adjacent node which are shifted due to abrupt change in location of centroidal axis
ELEMENTS	Input total number of elements in the complete structure
FRAME ELEMENTS	Input elements of the structural concrete frame
SPRINGS	Input springs and their stiffnesses
PRESTRESSING STEEL	Input prestressing steel properties
TENDON GEOMETRY	Define tendon numbers and geometry of each in the structure
TRAVELERS	Define traveling formwork
MESH COMPLETE	Signals end of structure definition
Set Ambient Conditions:	
SET	Define ambient conditions for the solution; such as day, temperature and solution technique parameters
Construction Operations:	
CHANGE STRUCTURE	Start construction operations for this step
BUILD	Install frame, spring, slave and hinge elements

RESTRAINTS	Change boundary conditions; also used to define <i>master</i> and <i>slave</i> constraints
(blank line in input)	
REMOVE	Remove frame, spring, slave and hinge elements
STRESS	Stress tendons
DE - STRESS	De-stress (remove) previously stressed tendons
MOVE	Move traveler to new destination nodes
CHANGE COMPLETE	End construction operations for this step
External Loading Commands: LOADING	Specify load increment which can consist of nodal loads, nodal displacements, and element temperatures
Solution Command: SOLVE	Solve the current structure
Output Commands: OUTPUT	Output total structure response
CAMBER	Output displacements to be adjusted for camber control
Problem Termination Command: STOP	Stop execution, save for restart

3.3 SAMPLE INPUT FOR A NONPRESTRESSED CANTILEVER BEAM

3.3.1 Description of the Structure

The following is sample input for a simple non-prestressed cantilever with uniform rectangular cross-section subject to a concentrated loading at its tip. For the purposes of illustration, the weight of the cantilever is specified as zero. Hence the actions (moments and shears) and the deformations will be due to the concentrated load only.

The cantilever is cast at day zero; loaded at day 100, at which time its moments and deformations are calculated. Without changing the loading, two other solutions are obtained, one at 1000 days and the other at 10,000 days. Since the loading is not changed, the actions will be the same for the different ages, but the deformations will increase. The change in deformation is triggered by the non-zero creep (Cr) and shrinkage (Sh) coefficients entered under “CONCRETE PARAMETERS” command. Cantilever selfweight (w) is not included.

The output of this example is given in Appendix A.

3.3.2 Input File

```

=====
; ADAPT-BRIDGE-INCREMENTAL  ABI      SOFTWARE MANUAL EXAMPLES
;=====
;   name of this file: CANT-MNL
;                               units are in lb-in
;

START
TITLE N=2
      CANT-MNL CANTILEVER BEAM EXAMPLE FOUR
      ELEMENTS FIXED AT LEFT UNITS lb-in
UNITS U=USA
CONCRETE PARAMETERS N=1 ! 1
MESH INPUT
NODES N=5
1 X=0 Y=0 ! 5 X=160 Y=0 G=1,5

CONCRETE PROPERTIES N=1
1 Fpc=5000 Cr=3.5 Sh=0.0005 W=0.0

```

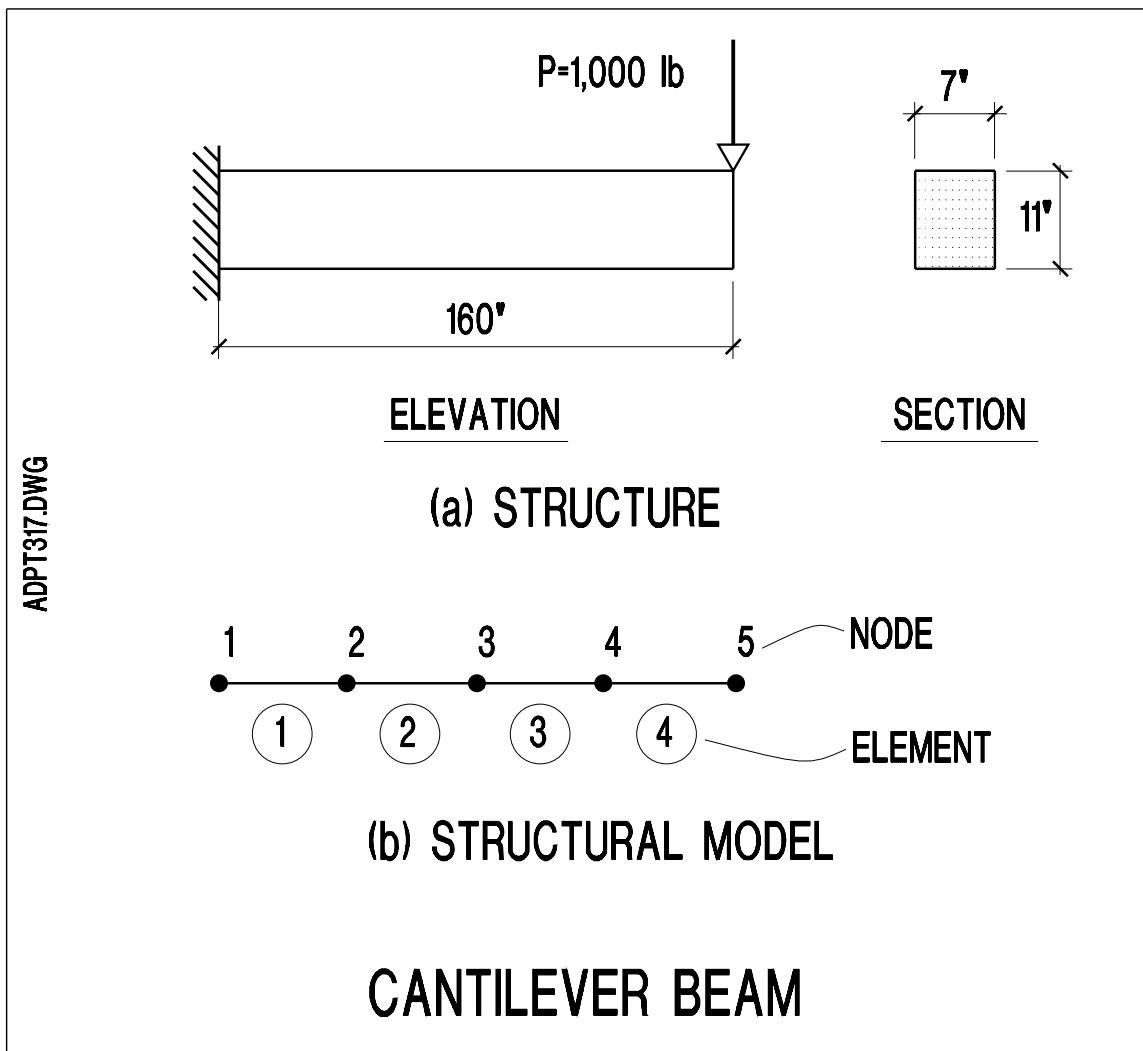


FIGURE 3.3-1

```
MILD STEEL PROPERTIES N=1
1 Es=29000000 P=0.05

SECTION PROPERTIES N=1      ! 1 D=11 B=7

ELEMENTS N=4
      FRAME N=4
1,1,2   C=1 X=1 St=1 Day=0 G=1,4,1,1,1

MESH COMPLETE

SET Day=0

CHANGE STRUCTURE

BUILD N=1,4,1
RESTRAINTS ! 1 R=1,1,1

CHANGE COMPLETE
LOADING
N=5 F=0, -1000,0

SOLVE      ! OUTPUT
SOLVE Day=7      ! OUTPUT
SOLVE Day=14     ! OUTPUT
SOLVE Day=28     ! OUTPUT
SOLVE Day=56     ! OUTPUT
SOLVE Day=100    ! OUTPUT
SOLVE Day=500    ! OUTPUT
SOLVE Day=2000   ! OUTPUT
SOLVE Day=10000  ! OUTPUT
STOP
```

3.4 SAMPLE INPUT FOR A PRESTRESSED CONCRETE BOX GIRDER BRIDGE

3.4.1 Description of the Structure

This example demonstrates ADAPT-ABI Command usage and the free field input format for a simple non-segmental bridge structure. Input listings for an example of a segmentally constructed bridge are given in Chapter 6.

A continuous prestressed concrete box girder bridge is shown in **Fig. 3.4-1**. The three span cast in place girder is simply supported at the first and last supports. It is rigidly connected to the columns at the second and third supports. The prismatic cross section is a single cell box with cantilevered top slab. Six post-tensioned prestressing tendons extending the full length of the bridge are draped in each web.

The structural model consists of 33 nodes, 32 frame elements and one prestressing tendon. The piers and girder are erected in one step on day 28 from concrete cast on day 0. After the initial analysis for dead load and prestressing, the time dependent structural behavior of the bridge is traced over a 10000 day (27 year) period.

3.4.2 Input File

```

=====
; ADAPT-BRIDGE-INCREMENTAL  ABI      SOFTWARE MANUAL EXAMPLES
=====
;   name of this file: EX1-MNL
;                               units are in lb-in

START      ;EX1-MNL

TITLE N=3
      EX1-MNL CONTINUOUS PRESTRESSED BRIDGE EXAMPLE
      SINGLE STEP CONSTRUCTION THEN 27 YEAR ANALYSIS
      ALL UNITS ARE POUNDS AND INCHES

UNITS U=USA
CONCRETE PARAMETERS N=1 ! 1 M=ACI

MESH INPUT

NODES N=33
      1 X=0      Y=870.59      ! 11 X=18000 Y=870.59 G=1,11
      21 X=4200 Y=870.59 G=11,21 ! 31 X=6000 Y=870.59 G=21,31
      32 X=1800 Y=0      ! 33 X=4200 Y=0

CONCRETE PROPERTIES N=1
      1 Fpc=5000 Cr=3.5 Sh=0.0005 W=155/1728

MILD STEEL PROPERTIES N=1 ! 1 Es=29000000 P=0.02

SECTION PROPERTIES N=2
      1 D=14,73,9 B=540,36,204 ! 2 D=96 B=180

ELEMENTS N=32
      FRAME N=32
      1,1,2      C=1 X=1 St=1 Day=0 G=1,30,1,1,1
      31,11,32 C=1 X=2 St=1 Day=0
      32,21,33 C=1 X=2 St=1 Day=0

PRESTRESSING STEEL N=1
      1 Ep=27000000 Meu=0.25 K=0.0004/12 Fpu=270000 R=10

TENDON GEOMETRY N=1
1 SPANS=3 M=1 AREA=11.54
1 B=0,900 E=10,900 N=11 G=1,11 ! R=0,0.5,0.1 S=-29.41,-84,-12
2 N=11 G=11,21! R=0.1,0.5,0.1 S=-12,-84,-12
3 N=11 G=21,31! R=0.1,0.5,0 S=-12,-84,-29.41

MESH COMPLETE

SET Day=28 G=0,-1

CHANGE STRUCTURE

```

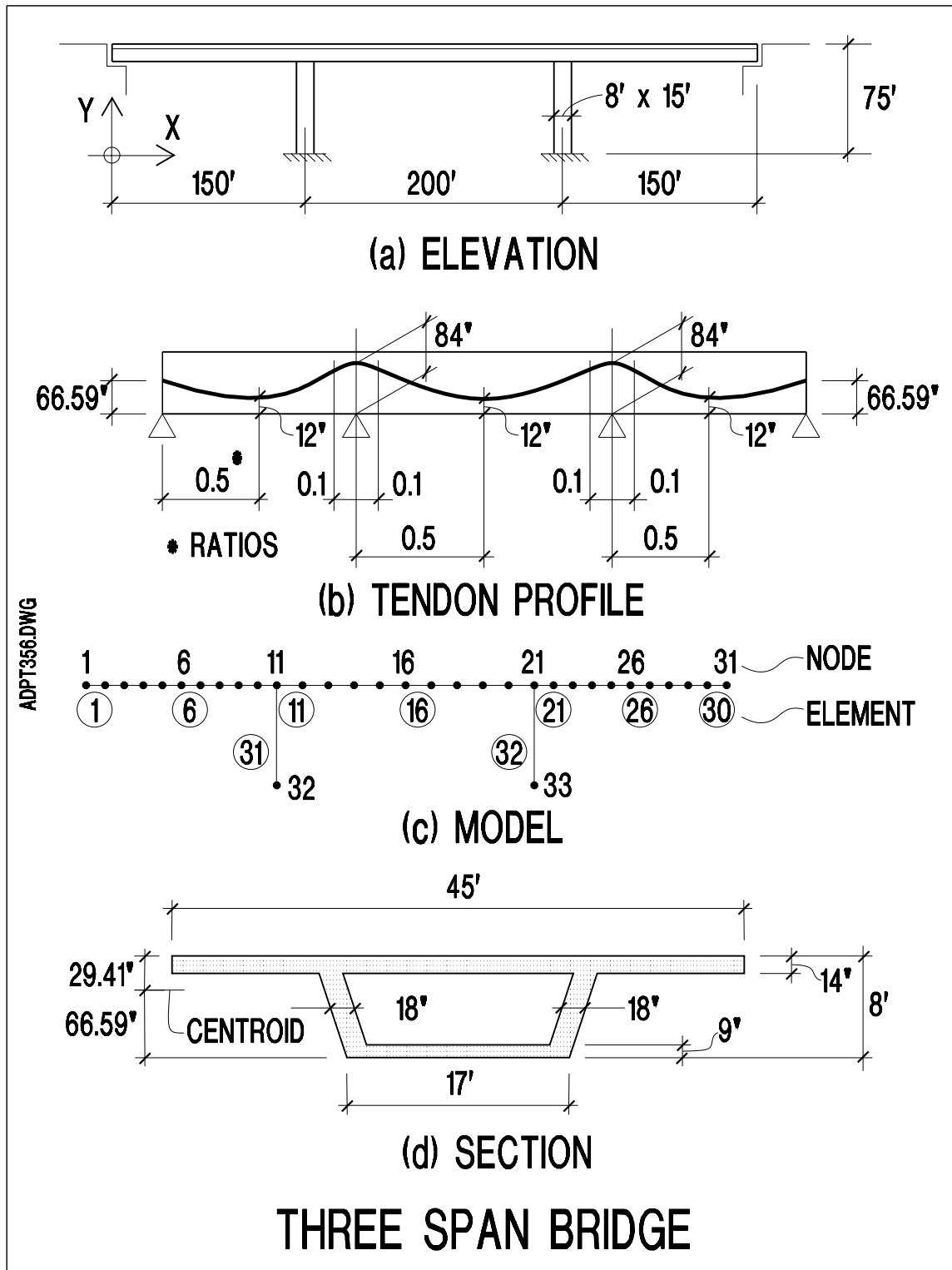


FIGURE 3.4-1

```

BUILD N=1, 32
STRESS N=1 F=2.5E5, 2.5E5
RESTRAINTS
1, 31, 30 R=0, 1, 0
32, 33 R=1, 1, 1

```

```
CHANGE COMPLETE
```

```
SOLVE                ! OUTPUT
SOLVE Day=50         ! OUTPUT
SOLVE Day=500        ! OUTPUT
SOLVE Day=1000       ! OUTPUT
SOLVE Day=5000       ! OUTPUT
SOLVE Day=10000      ! OUTPUT
STOP
```

3.5 DETAILED DESCRIPTION OF COMMAND SYNTAX

The detailed syntax of each command and a description of its use, actions and output are described. The commands are organized alphabetically. A sample sequence in which the commands normally will be found in a typical input file is given in Section 3.2.1. Most commands are optional, and must not be used unless required for the structure under analysis. For example, a command such as TRAVELERS N=0 need not be used as part of input data if no travelers are present.

Default values for input quantities, where implemented in the program, are indicated within square brackets by [?] in the descriptions. [pv] indicates a default value assumed to be equal to the previous value entered.

ACTIVATE EXTRACT

Syntax: ACTIVATE EXTRACT

Explanation: When included in the data file, this command will cause the program (ADAPT-ABI) to extract the components of the dead loading and the long-term effects of creep and shrinkage incrementally with the progress of computations in ADAPT-ABI. It generates a set of data which will be required once ABI-Gen is invoked. Its location is anywhere on a line after the START command and before MESH INPUT.

Example: START

ACTIVATE EXTRACT

Purpose: The ACTIVATE EXTRACT does not change the results of ABI module, but makes it somewhat slower to execute, since it generates additional data for the module ABI-Gen.

BUILD

Syntax: BUILD N=n1,n2,inc Day=?

where,

- n1 = Element number of first frame element in a series of elements to be installed in the structure
- n2 = Element number of last element in the series [n1]
- inc = Element number increment
- Day = Casting date of concrete in the specified elements

Explanation: The BUILD command is used to install new frame elements into the structure. The elements which are installed can later be removed using the REMOVE subcommand. The sequence of elements generated with the N= identifier must be statically feasible in order for node displacement initialization to work properly. Thus, for the BUILD command, backwards generation (i.e. inc less than 0) is allowed. If a casting date is specified under this command, it overrides the value input under the FRAME ELEMENTS subcommand of the MESH INPUT command.

The dead load of the frame element is automatically included as concentrated forces at the nodes based on the length, cross section area and material unit weight of the elements input under the MESH INPUT command, multiplied by the current gravity load multipliers specified with the SET command.

The displacements of any previously unrestrained nodes which as a result of this command are made active, are initialized based on the total displacements of the node at the other end of the element and assumed rigid behavior of the element added. This makes it necessary under this subcommand to generate elements in a statically feasible order.

More than one Build command can be specified in order to achieve a construction.

Note: n2, inc, and, Day=?, are optional.

Example: BUILD N=12
BUILD N=10, 8, -2 DAY=32

CAMBER

Syntax: CAMBER C=?

where,

- C = Displacement component to be printed [2]
= 1 for X- displacement

= 2 for Y- displacement
= 3 for Z- rotation

Explanation: The CAMBER command is used to compute and print displacement adjustment for all nodes in the structure. Each time the OUTPUT command is used, a record of total nodal displacements is saved. Under the CAMBER command, the current total nodal displacements are subtracted from the position of each node at the time of installation of that node. The significance of the output camber values is as follows. If each node be positioned at time of its installation with the amount of camber output by the program, the structure will have zero total displacement at the time and condition associated with the call of camber command. Camber command can be called several times. Each camber solution refers to the associated time and construction phase of the *camber* call. The background to camber calculations is given in appendices of this manual in greater detail.

The graphical output displays the required camber each time the OUTPUT command is invoked. However, a tabular printout of the nodal camber is generated only if the camber command is called.

The graphical output displays the vertical and horizontal components of the camber, whereas the tabulated output lists all the three x- y- and rotation displacements.

CHANGE COMPLETE

Syntax: CHANGE COMPLETE

Explanation: The CHANGE COMPLETE command has no arguments. It is the closing companion to CHANGE STRUCTURE command. It signals the program that the interpretation of CHANGE STRUCTURE commands should stop. The effects of the changes on the displacements and internal stresses in the structure are found by using the SOLVE command.

CHANGE STRUCTURE

Syntax: CHANGE STRUCTURE

Explanation: The CHANGE STRUCTURE command has no arguments of its own, but is followed by a series of construction operation subcommands which specify the current boundary conditions, the installation and removal of frame elements and prestressing tendons, and the movement of travelers in the plane frame structure. The structure to be analyzed is completely defined by the cumulative effects of CHANGE STRUCTURE commands.

The construction operation includes one or more subcommands nested between the two commands `CHANGE STRUCTURE` and `CHANGE COMPLETE`.

```
CHANGE STRUCTURE
  RESTRAINTS : to change nodal boundary conditions
  BUILD      : to install frame elements
  REMOVE     : to remove frame elements
  STRESS     : to stress, or to restress, prestressing tendons
  DE-STRESS  : to remove prestressing tendons
  MOVE       : to move traveling formwork
CHANGE COMPLETE : (to terminate the command) described before
```

The `CHANGE STRUCTURE` command can be used any number of times in a given analysis in order to model the construction sequence. The effects of the changes on the displacements and internal stresses in the structure are found by using the `SOLVE` command after the `CHANGE STRUCTURE` command and any optional loadings commands.

CONCRETE PARAMETERS

Explanation: The `CONCRETE PARAMETERS` command generates or concludes with either program defined tables of shrinkage strain, elastic modulus and creep coefficients vs. time or user defined values. The `CONCRETE PARAMETERS` command **must** be used prior to the `MESH INPUT` command.

These tables provide the constitutive constants used in the time dependent analysis of the structure. When automatic generation of the program is used, the values in the tables are normalized for ultimate creep coefficient =1, ultimated shrinkage strain =1 and concrete strength $f'_c(28) = 1$. Prior to use in each solution step, these normalized values are scaled by the creep coefficient, and shrinkage strain values input under the `CONCRETE PROPERTIES` subcommand of the `MESH INPUT` command. The correction of concrete strength f'_c is also achieved through scaling to f'_c given under the `CONCRETE PROPERTIES` subcommand. Thus when automatic generation is used, under most circumstances only one concrete parameter type is required.

Time dependant concrete behavior can be modeled according to a number of model codes, laboratory test data, or user defined values. Each parameter type can employ a different model.

The `CONCRETE PARAMETERS` details the specifics of the concrete material models used. They generate data for shrinkage strain, elastic modulus and creep coefficients vs. time. The `CONCRETE PARAMETERS` command **must** be used prior to the `MESH INPUT` command. More than one “concrete material

model” can be specified. The material models will be specified sequentially one after the other.

Use either (i) the *short form* input, which takes advantage of default values built in the program, or (ii) *detailed input*, which involves a comprehensive definition of concrete properties.

(i) SHORT FORM INPUT

Syntax: CONCRETE PARAMETERS N=n
n M=model

n = Number of concrete material models

Explanation: The CONCRETE PARAMETERS specifies the number of different concrete types used (from shrinkage and creep modeling standpoint)

where,

n = Parameter type number
model = Model type used [ACI]
= ACI for ACI recommendations
= CEB for CEB/FIP recommendations
= CEB1 for one component CEB/FIP model

In ACI-209 (1978), the loading and unloading curves for creep follow the same pattern. In CEB-78 the unloading curve is different in shape to the loading curve. Over a long period, the unloading curve tends to pick up strain, contrary to observation. To improve the CEB model, CEB1 is introduced. The CEB1 model has the same unloading curve characteristic as the loading curve.

Fig. 3.5-1 illustrates the point.

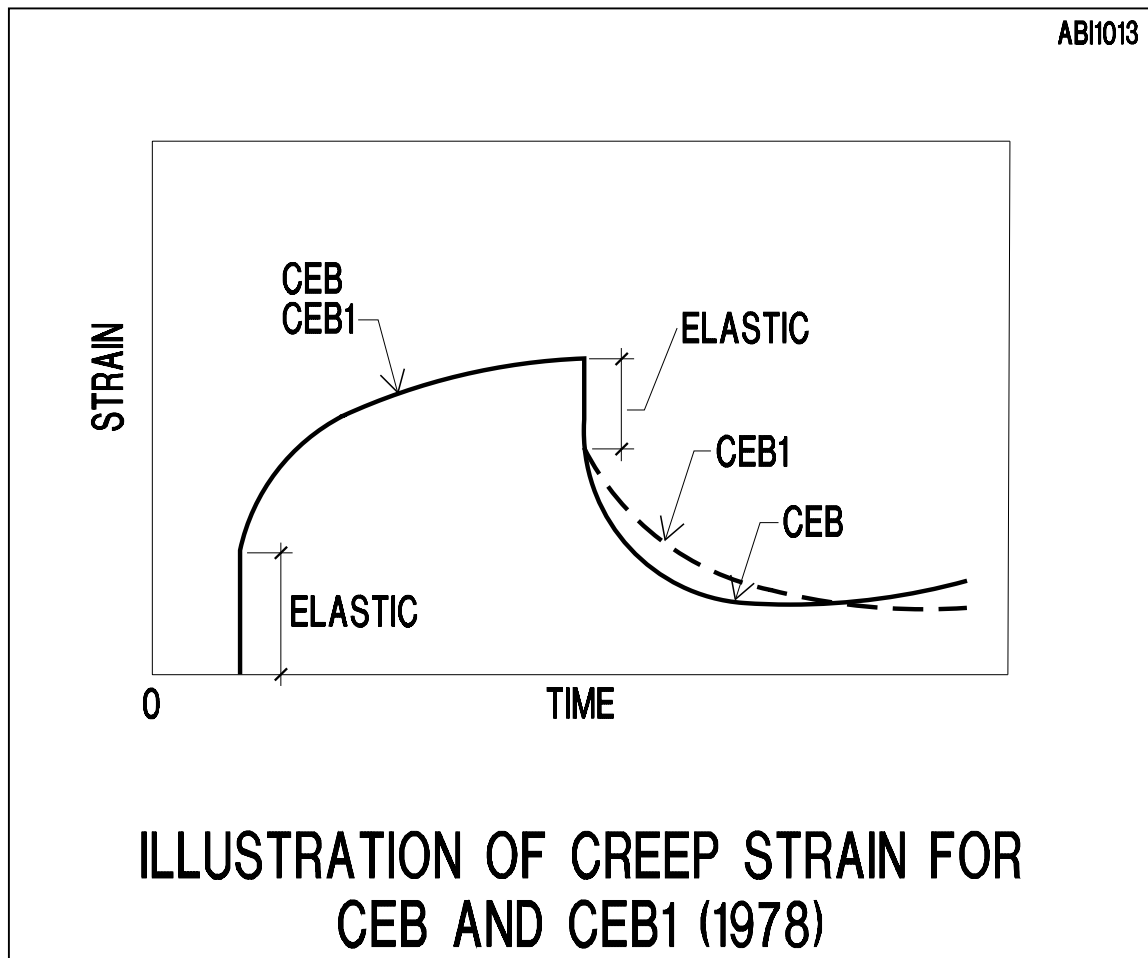


FIGURE 3.5-1

Example 1:

CONCRETE PARAMETERS N=1
1 M=ACI

This example uses ACI model for the entire frame with ACI-209 default parameters.

Example 2:

CONCRETE PARAMETERS N=2
1 M=ACI
2 M=CEB

This example uses ACI model for concrete *type 1* and, CEB for concrete *type 2*.

(ii) DETAILED INPUT

The syntax of the detailed input depends on the concrete model used. The following models are available:

- ACI-209 (1978)
- CEB/FIP; and CEB1/FIB (1978)
- LAB

The input option LAB refers to input from laboratory generated readings, or user generated values based on model codes different from those available in ABI library (ACI, and CEB).

A. ACI-209 Concrete Model (1978)

Syntax: CONCRETE PARAMETERS N=? T=na, nt
To be followed by several lines of concrete parameter specifications.

where,

- N = Total number of concrete parameter types;
- na = Maximum *loading ages* in parameter tables (number of creep curves) [32]; and,
- nt = Maximum number of data points on a single creep curve *after loading* for parameter tables [32].

For example in **Fig. 3.5-2** the collection of creep curves shown consists of creep curves for four loading ages 3,7,14 and 28 days. Using this set of data, the program can determine the creep response of a structure loaded between days 3 and 28, but observed from day one on to the farthest extent of the curves (typically 20 years).

The default of the program is 32 loading ages for the parameter na. The extent of the loading ages stretches up to 20 years. This enables determination of creep for loading applied essentially anytime from casting day to 20 years of age.

The number of points used to generate the creep curve in **Fig. 3.5-3** is six (nt=6). These points are either generated used model codes or are read from laboratory tests.

The next data line for each ACI concrete type model is a control data line which takes the following form. If the internally generated loading ages and observation times are used, this is the only line required for parameter type.

M=ACI T= nat, ntt

where,

- n = Concrete model number;
- nat = Number of loading ages in material type, n, table [na]. This parameter is for creep curves shown in **Fig. 3.5-2**; and
- ntt = Number of data after loading for material type, n, table [nt] This parameter is 6 for the creep curve shown in **Fig. 3.5-3**.

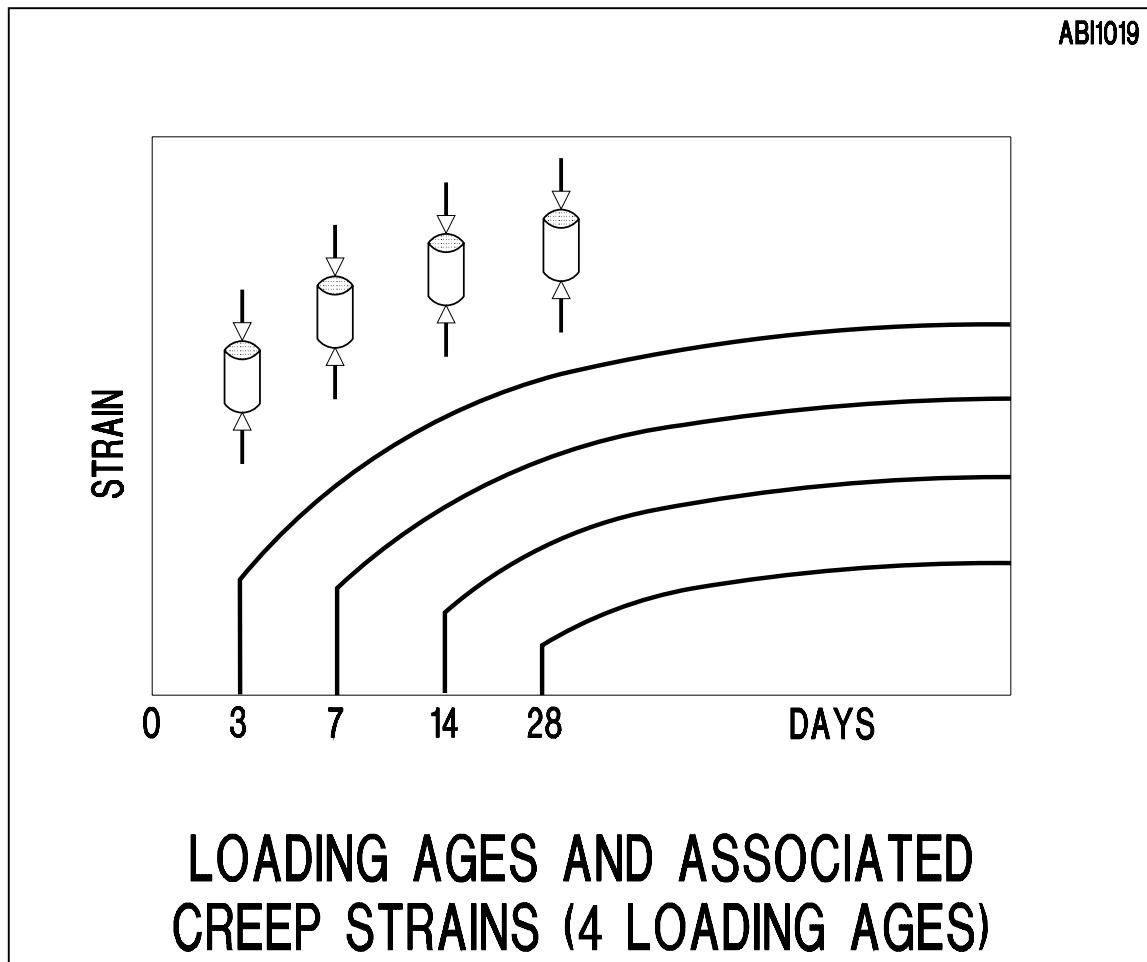


FIGURE 3.5-2

Example: 2 M=ACI

This instruction means that the second material type selected uses ACI model. Since τ is not specified, it uses ages and times coded in the program. It uses the maximum number of loading ages and data generation points for each creep curve.

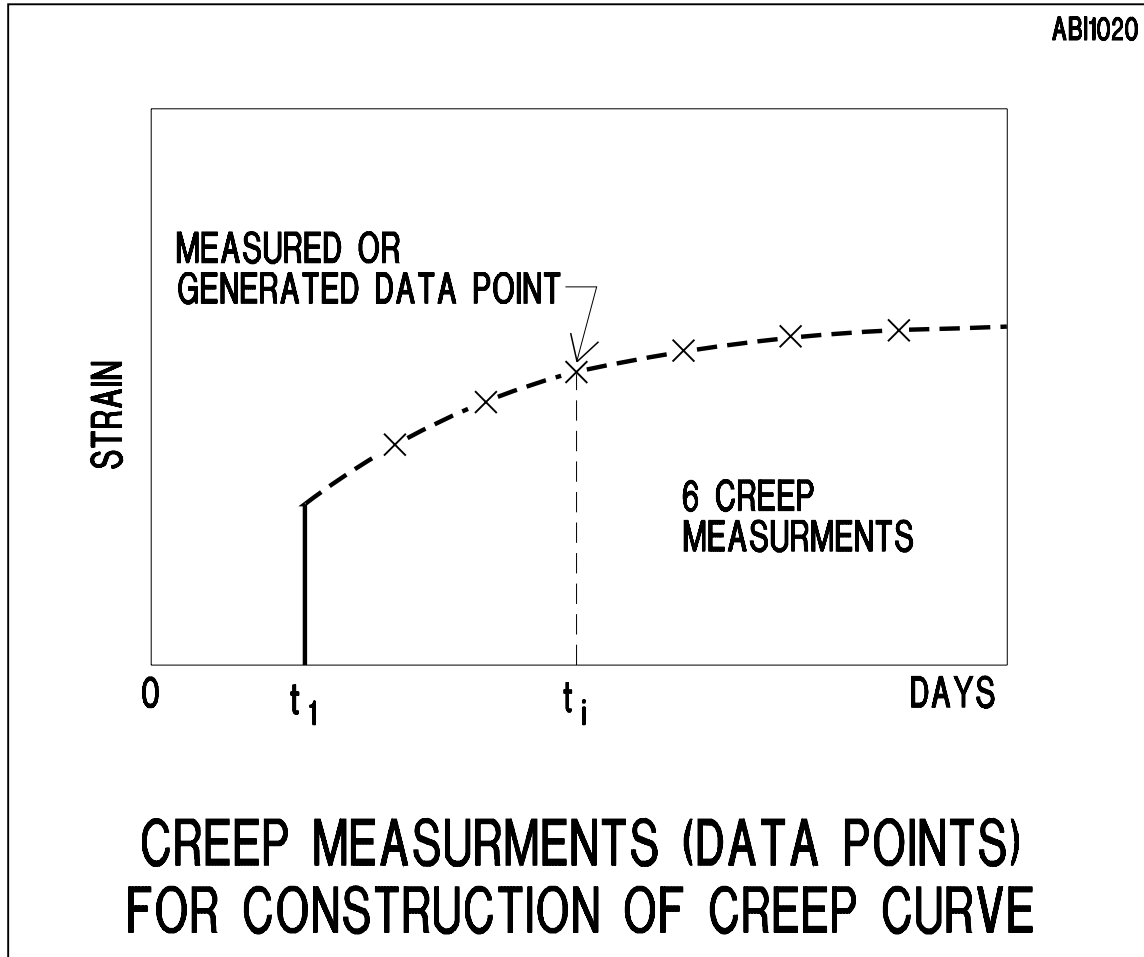
M=ACI

In addition to the optional τ parameter described before, eight other parameters may be described on the same line as "n m=ACI..." in order to fine tune the ACI concrete model to a user defined alternative. These parameters are:

a=? b=? c=? d=? e=? f=? t0=? W=?

All the eight preceding parameters are optional. User may specify as many of these as necessary after the two parameters of "n M=ACI."

where,

**FIGURE 3.5-3**

$a[4]$ and $b[0.85]$ are used in the time function for $f'_c(t)$ (see appendices for more detail):

$$f'_c(t) = f'_c(28)[t/(a+b*t)]$$

$c[1.25]$ and $d[0.118]$ are used in the age function for creep (see appendices for more detail):

$$K_\tau = c \cdot \tau^{-d}$$

$e[1]$, $f[50]$, and $t_0[7]$ are used in the time function for shrinkage:

$$\epsilon^s(t) = \epsilon^s(u)(t-t_0)^e/[f+(t-t_0)^e]$$

where,

$\epsilon^s(t)$ is shrinkage on day (t)

$\epsilon^s(u)$ is the ultimate shrinkage value

$W[150/(12)^3]$ is the unit weight of concrete in pounds per cubic inch. When using lb-in (American units) is used. In SI and MKS units the defaults are given. The general ACI formula for elastic module of elasticity is:

$$E(t) = 33 W^{1.5} [f'_c(t)]^{1/2}$$

where,

W is in pounds per cubic foot;

$f'_c(t)$ is in psi; and

E(t) is in psi.

In the three systems of units adopted in the program, the relationship assumes the following forms:

In American units:

$$E(t) = 2.3704 \times 10^6 W^{1.5} [f'_c(t)]^{1/2}$$

where,

W is in pounds per cubic inch $[8.6806 \times 10^{-2}]$;

$f'_c(t)$ is in psi; and

E(t) is in psi.

In SI units:

$$E(t) = 1.3518 \times 10^{12} W^{1.5} [f'_c(t)]^{1/2}$$

where,

W is in kg/mm^3 $[2.4019 \times 10^{-6}]$;

$f'_c(t)$ is in N/mm^2 ; and

E(t) is in N/mm^2 .

In MKS units:

$$E(t) = 1.3648 \times 10^8 W^{1.5} [f'_c(t)]^{1/2}$$

where,

W is in kg/cm^3 $[2.4019 \times 10^{-3}]$;

$f'_c(t)$ is in kg/cm^2 ; and

E(t) is in kg/cm^2 .

B. CEB/FIP; CEB1/FIP

The next data line for each CEB or CEB1 concrete type model is a control data line which takes the following form. If the internally generated loading ages and observation times are used, this is the only line required for the parameter type.

n Model [Area=? P=? H=?]

where,

n = Parameter model number;
 Model = CEB for CEB/FIP recommendations;
 = CEB1 for one component CEB/FIP model;
 H = ambient relative humidity (percent) [70];
 Area = Area of concrete cross-section [300 cm², 30,000 mm², 46.50 in²]; and,
 P = Perimeter of concrete section in contact with the atmosphere [10 cm, 100 mm, 3.937 in].

Modulus of Elasticity:

ADAPT-ABI uses the following relationship to determine the modulus of elasticity when CEB or FIB concrete models are selected.

$$E_c(t) = 45680[f'_c(t)]^{1/2}$$

where,

$E_c(t)$ = modulus of elasticity on day t, in N/cm²; and,
 $f'_c(t)$ = concrete cylinder strength on day t, in N/cm².

Aging of Concrete

Aging in concrete is effected through change in concrete strength and thereby modulus of elasticity. The change in modulus of elasticity with time is effected using the following relationship.

$$f'_c(t) = [(1.45*t^{0.75})/(t^{0.75} + 5.5)]*f'_c(28)$$

where,

$f'_c(t)$ = concrete strength on day, t; and,
 $f'_c(28)$ = concrete strength on day 28.

Note: Equation B.2.5 of ADAPT-ABI manual refers to concrete strength at day=1000. For modulus of elasticity at other days, the curve in **Figure B.2.2(a)** can be used. The two relationships given herein combine the equation B.2.5 of the manual and the **Figure B.2.2(a)**.

C. LAB

The input for laboratory generated information, or for codes not programmed in the library of ADAPT-ABI, the LAB option of input data is used. In this option the user inputs the shrinkage, creep and change of modulus of elasticity of concrete with time. More than one concrete material model can be defined.

Syntax: CONCRETE PARAMETERS N=? T=n1,n2 MaxShrinkageReadings=?
n M=LAB CreepSpecimens=? CreepReadings=? ShrinkageReadings=?

```

Age          ShrStrain          ;Shrinkage header
- - - -      - - - - - -      ;Shrinkage data
- - - -      - - - - - -      ;      "      "
- - - -      - - - - - -      ;      "      "

LoadingAge=? Eci=?              ;Creep curve identification

ObservationAge  CreepStrain      ;Creep curve header
- - - -      - - - - - -      ;Creep curve data
- - - -      - - - - - -      ;      "      "      "
- - - -      - - - - - -      ;      "      "      "

```

where,

N	= Total number of concrete material models specified;
n1	= Largest number of creep specimens tested for any material model;
n2	= Max. number of creep measurements taken on a specimen;
MaxShrinkageReadings	= Largest number of shrinkage readings used for any material model;
n	= Number of the concrete material model;
CreepSpecimens	= The number of creep specimens tested for the current material model. Can be less than n1 above;
CreepReadings	= The largest number of creep table readings allowed for each one of the creep specimens of the current material model. Different material models and different specimens in the same model can have different numbers of readings. Can be less than n2 above;
ShrinkageReadings	= Number of data readings for the shrinkage test; Can be less than MaxShrinkageReadings above;
Age	= Age of concrete in days when shrinkage readings are taken. Minimum age is 1 day;
ShrStrain	= Shrinkage strain readings from the lab specimen;

LoadingAge	= Age of the lab specimen, in days, when the creep loading is applied. Minimum age is 1 day;
Eci	= Modulus of elasticity of the lab specimen at the time the creep loading is applied; determine from the instantaneous deformation of lab specimen under applied loading;
ObservationAge	= Days when measurment of the lab creep specimen are made; minimum age is 1 day; and,
CreepStrain	= Observed creep strain of the lab specimen per unit stress; e.g. strain (per Mpa for SI units; per psi for Ameican units; per Kg/cm ² for MKS units).

Explanation: The above syntax for the CONCRETE PARAMETERS command is a modification of the syntax used in previous releases of the program for the laboratory generated material model option (M=LAB). In this case the shrinkage, creep and elastic modulus data to be used in program calculations are input by the user. The number of different material models and the number of creep and shrinkage test specimens for each material are user defined. The number of ShrStrain readings must be equal to the number specified for ShrinkageReadings. The number of CreepStrain readings must be equal to the number specified for CreepReadings.

The program then uses the input data to interpolate solutions for days not specifically covered by the input. Note that the program cannot extrapolate from the user specified data beyond the final day of shrinkage strain data collection. That is to say, if the final reading for shrinkage strain data is taken on day 45; and for creep strain on day 50, then the program will only be able to find a solution for days up to and including day 45.

Example:

```

CONCRETE PARAMETERS N=2 T=4,8 MaxShrinkageReadings=8
1 M=LAB CreepSpecimens=2 CreepReadings=6 ShrinkageReadings=7
  Age          ShrStrain
  1            0E-6
  5            29E-6
  10           50E-6
  15           80E-6
  20           79E-6
  30           86E-6
  40           111E-6      ; note there are 7 entries

LoadingAge=1 Eci=10894      ; first creep specimen
ObservationAge      CreepStrain
  1                696E-8
  5                1581E-8

```

```

          10          1842E-8
          15          2074E-8
          20          2161E-8
          30          3306E-8      ; 6 readings

LoadingAge=5 Eci=18684          ; second creep specimen
      ObservationAge      CreepStrain
          5          493E-8
          10          870E-8
          15          1117E-8
          20          1305E-8
          25          1410E-8
          30          1523E-8      ; 6 readings

2 M=LAB CreepSpecimens=1 CreepReadings=5 ShrinkageReadings=6
      Age      ShrStrain
          1          0E-6*2
          7          29E-6*2
          14          50E-6*2
          21          80E-6*2
          28          79E-6*2
          35          86E-6*2      ; 6 readings

LoadingAge=3 Eci=16203
      ObservationAge      CreepStrain
          3          566E-8
          7          1291E-8
          11          1682E-8
          22          2320E-8
          32          2683E-8      ; 5 readings

```

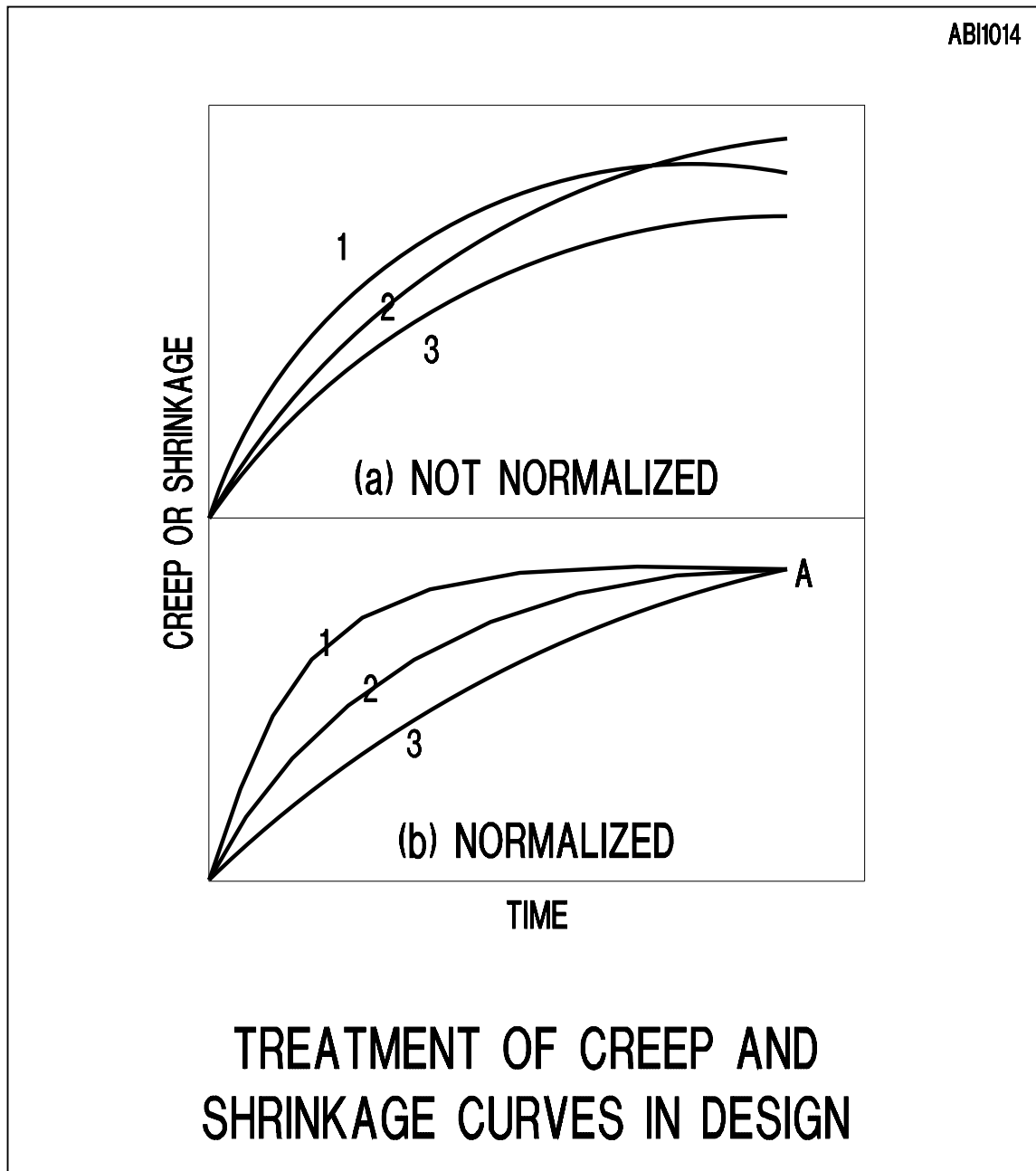
CONCRETE PROPERTIES

Syntax: CONCRETE PROPERTIES N=?
 n Fpc=f'c Cr=? Sh=? W=? M=? Ac=?

where,

N = Total number of concrete types [1];
 n = Concrete type number;
 Fpc = Ultimate compressive strength (f'_c) cylinder 28 days;
 Cr = Ultimate creep coefficient [0];
 Sh = Ultimate shrinkage coefficient [0];
 W = Unit weight (for computing gravity dead load only give units in kg/mm³. This is not used in the calculation of modulus of elasticity.) [0];
 M = Time dependent concrete material model number [1]; and,
 Ac = Thermal expansion coefficient [0].

Explanation: The CONCRETE PROPERTIES command is used to specify the different basic concrete material properties found in the structure. The concrete type number n must be less than or equal to the total number of concrete types input on the

**FIGURE 3.5-4**

CONCRETE PROPERTIES command line. The concrete types may be supplied in any order; however each concrete type must be specified once.

The definition of the parameters C_r and S_h are further clarified through **Fig. 3.5-4 (a)** shows symbolically the variation of either coefficient with time for three materials identified as 1, 2, and 3. The maximum ordinate of each curve gives the ultimate coefficient (say C_r). To increase the generality of the program, the concrete models, such as the materials 1, 2 and 3 shown in the figure are defined under the subcommand CONCRETE PARAMETERS. Under CONCRETE PROPERTIES subcommand, the ultimate coefficient, will reach the

specified value at time infinity, but will do so following different paths (1, 2 and 3 shown in part (b) of the figure).

Example: (in SI units)

```
CONCRETE PROPERTIES N=1
  1 Fpc=36 Cr=2.5 Sh=0.0004 W=2.4x10-6
```

Where the maximum creep strain of a standard specimen for this material is 2.5. The selection of the code and concrete material for the standard creep specimen is given under the command CONCRETE PROPERTIES. The value Cr=2.5 identifies the location of point A in **Fig 3.5-4(b)**, but does not specify which of the three curves 1,2, or 3 are to be used. This is done in the CONCRETE PROPERTIES command. Like-wise the maximum value of shrinkage strain for a standard specimen for the concrete material used is Shr=0.0004. The weight in kg/mm³ is given by w.

Example:

```
CONCRETE PROPERTIES N=2
  1 Fpc=30 Cr=1.0 Shr=1.0 W=0 M=1
  2 Fpc=20 Cr=1.0 Sh=1.0 W=2.4x10-6 M=2
```

This example applies to when the shrinkage and creep are specified using at laboratory or a model code not included in ADAPT-ABI library. Since the entire variation as shown in **Fig. 3.5-4(b)** is given in the input tables of CONCRETE PARAMETERS command, no normalization is used.

DE - STRESS

Syntax: DE-STRESS N=?
List=n1,n2 ... n G=n1, n2, inc

where,

N	= Total number of tendons [1];
List	= List of tendons to be de-stressed;
G	= This optional parameter generates the list of tendons to be de-stressed;
n1, n2, inc	= Generation parameters;
n1	= Tendon number for start of generation;
n2	= Last tendon number in generation; and,
inc	= Generation increment

Explanation: This command is used to remove tendons previously stressed. The removal of tendon will result in its stiffness being eliminated from the stiffness matrix of the

structure. If a tendon is being re-stressed to a different stress level, the STRESS command must be used.

Example:

```
DE-STRESS  N=5
LIST=2, 4, 6, 11, 14

DE-STRESS  N=5
G=2, 10, 2
```

ELEMENTS

Syntax: ELEMENTS N=?

where,

N = Total number of frame elements destined to be used in the structure during the construction, or when it is complete. N is the sum of the following elements currently available in the software's library:

FRAME elements
SPRING elements

The TRAVELER elements are **not** included herein. Stay elements are handled under prestressing tendons.

FRAME

Syntax: FRAME N=?
 n, ni, nj C=? X=?,? St=? Day=? Off=?
 [G=n1, ..., n11]

where,

N = Total number of frame elements [1];
n = Frame element number ($n \leq N$);
ni = Node number for node I;
nj = Node number for node J;
C = Concrete type number;
X = Section type number at node I and J. If only one number is entered, section is assumed to be uniform
St = Mild Steel type number;
Day = Casting Date (days);
Off = Offset entry number (**Fig. 3.5-5, and 3.5-6**) [0]; and,
n1 - n11 = Frame element generation parameters, optional as described below.

Explanation: The FRAME ELEMENTS command is used to define all the frame elements used in modeling the structure. The frame element number n must be less than or equal to the total number of frame elements input on the FRAME ELEMENTS command line. The frame element descriptions may be supplied in any order; however each frame element description must be specified or generated once.

Frame elements (**Figs. 3.5-7**) consist of parallel concrete and mild steel components. Data supplied with the C= and X= identifiers specifies the concrete component. Data supplied with the St= and X= identifiers specifies the uniformly distributed mild steel component. The casting date (in days) may be specified with the Day= identifier, but this specification can be overridden under the BUILD subcommand of the CHANGE STRUCTURE command.

The positive direction of the local coordinated system of an element is from its node I to its node J (**Fig. 3.5-7**). Node I is the node entered first in the input data under the ELEMENTS command. The top and bottom fibers of an element are defined by the orientation of its local coordinate system as shown in **Fig 3.5-7**.

More than one frame element may be generated with different Off= subcommands at an analysis node. This will result in elements to be located at different physical locations with respect to the reference nodes (**Fig. 3.5-6**).

Elements with different cross sections at their ends (I and J) are idealized as a prismatic uniform section. The idealization is for mid-length of the element, and it is obtained through linear interpolation of values entered for the ends I and J.

The idealized section is used for stiffness and time- dependent computations. However, at completion of the computation the actions at nodes I and J are applied to the actual section properties entered for I and J to determine fiber stresses in concrete (**Fig. 3.5-8**).

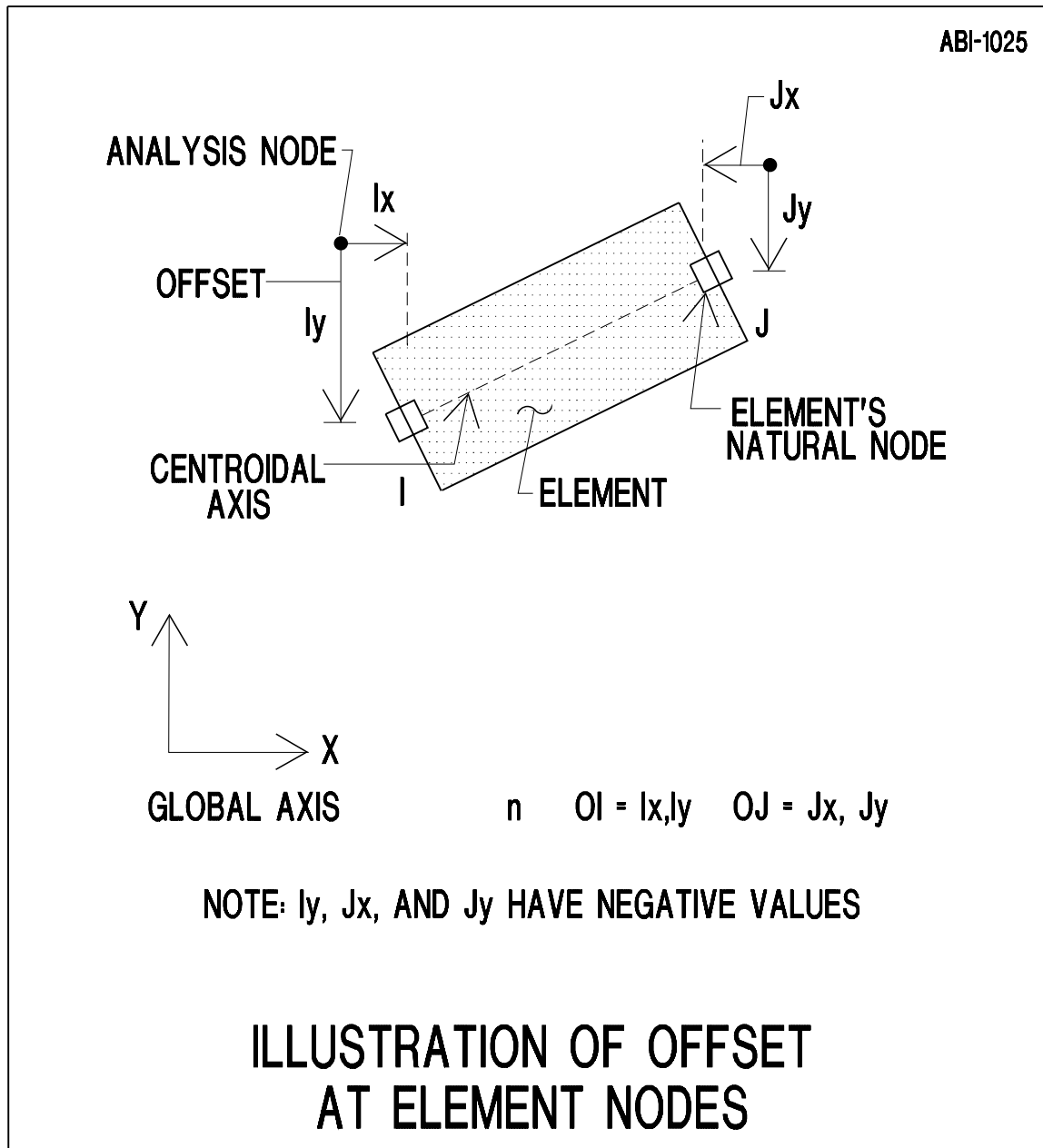


FIGURE 3.5-5

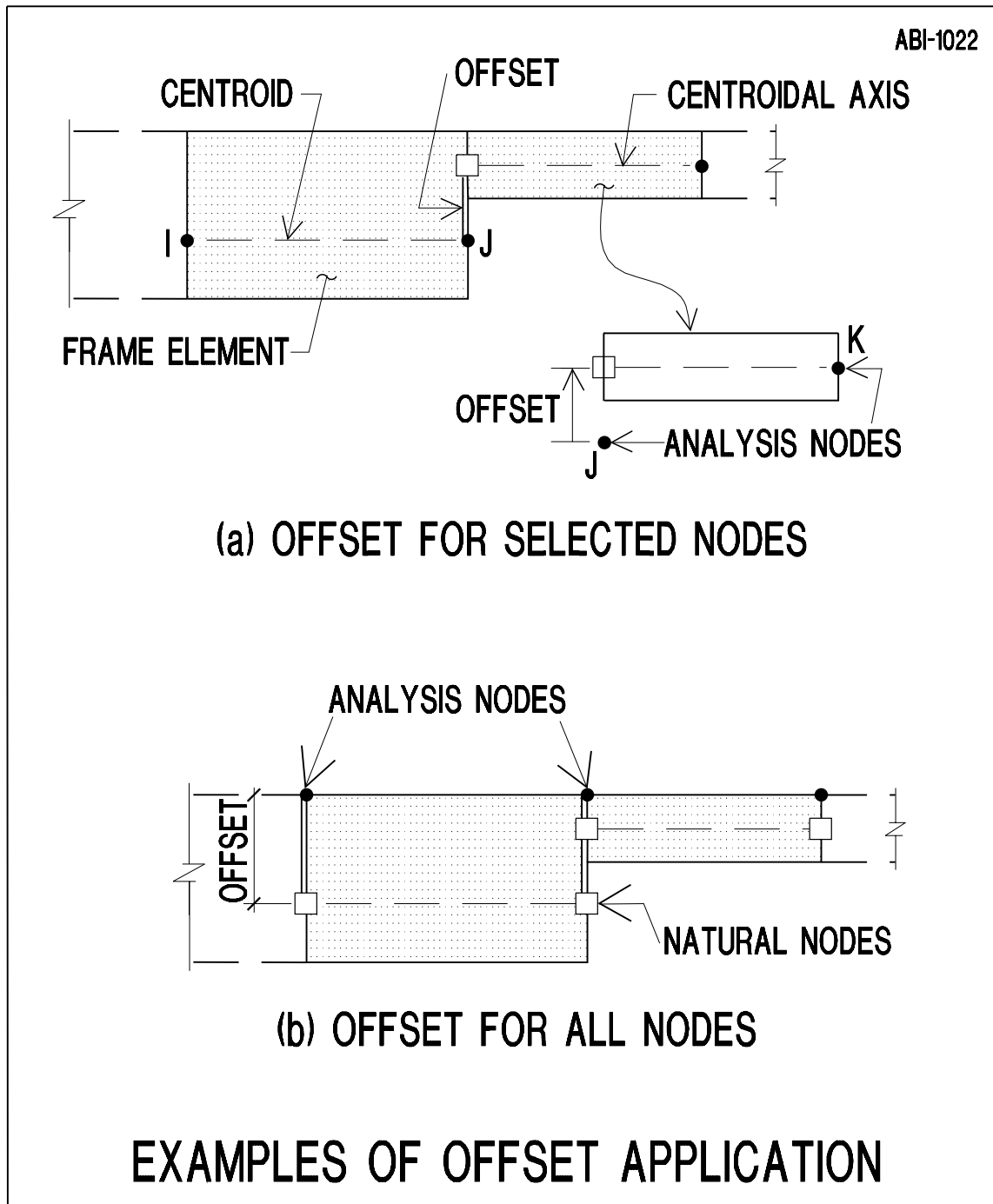
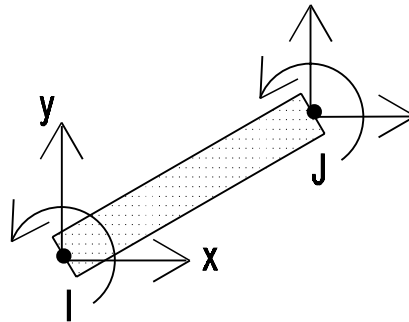
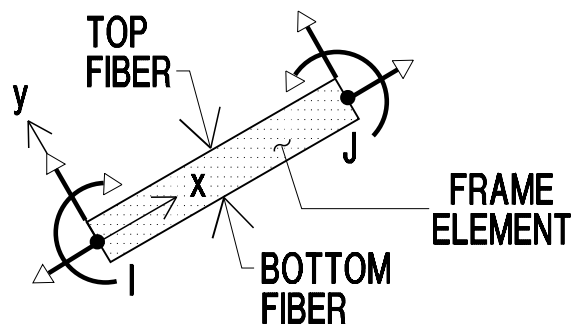


FIGURE 3.5-6

ABI1021



(a) GLOBAL SYSTEM AND DIRECTION OF
POSITIVE GLOBAL ACTIONS



(b) LOCAL SYSTEM (X,Y) AND DIRECTION OF
POSITIVE LOCAL ACTIONS

ELEMENT COORDINATES

FIGURE 3.5-7

Additional frame elements may be automatically generated using the G=n1, . . . , n11 parameters. Frame elements are generated by incrementing the input parameters (above) by their respective increments, which are input using the G= identifier. The generation parameters are defined as:

n1 = First element in generation sequence;
 n2 = Last element in generation sequence;
 n3 = Frame element number increment;
 n4 = Node I increment;
 n5 = Node J increment;
 n6 = Concrete type increment;
 n7 = Cross section type increment at node I;
 n8 = Cross section type increment at node J;
 n9 = Offset data increment;
 n10 = Mild steel type increment; and,
 n11 = Casting date increment.

This sequence of lines must be terminated by a blank line.

Example:

```
ELEMENTS N=16
FRAME N=16
  1,1,2    C=1 X=1 St=1 Off=1 DAY=0  G=1,8,1,1,1
  . . . .
  9,1,2    C=1 X=2 St=1 Off=2 DAY=0  G=9,16,1,1,1
  . . . .
```

LOADING

Syntax: LOADING

To be followed by one or more lines of loading data input.

Explanation: The LOADING command is used to apply concentrated nodal loads, imposed nodal displacements, uniformly distributed frame element loads and temperature change loadings to the structure. Any node or element to which a loading is applied must be a part of the current structure. All loadings remain in effect until they are removed by the application of an equal but opposite loading.

Each loading data line takes one of several forms depending on the type of loading. The loading data lines can contain all three forms in any order.

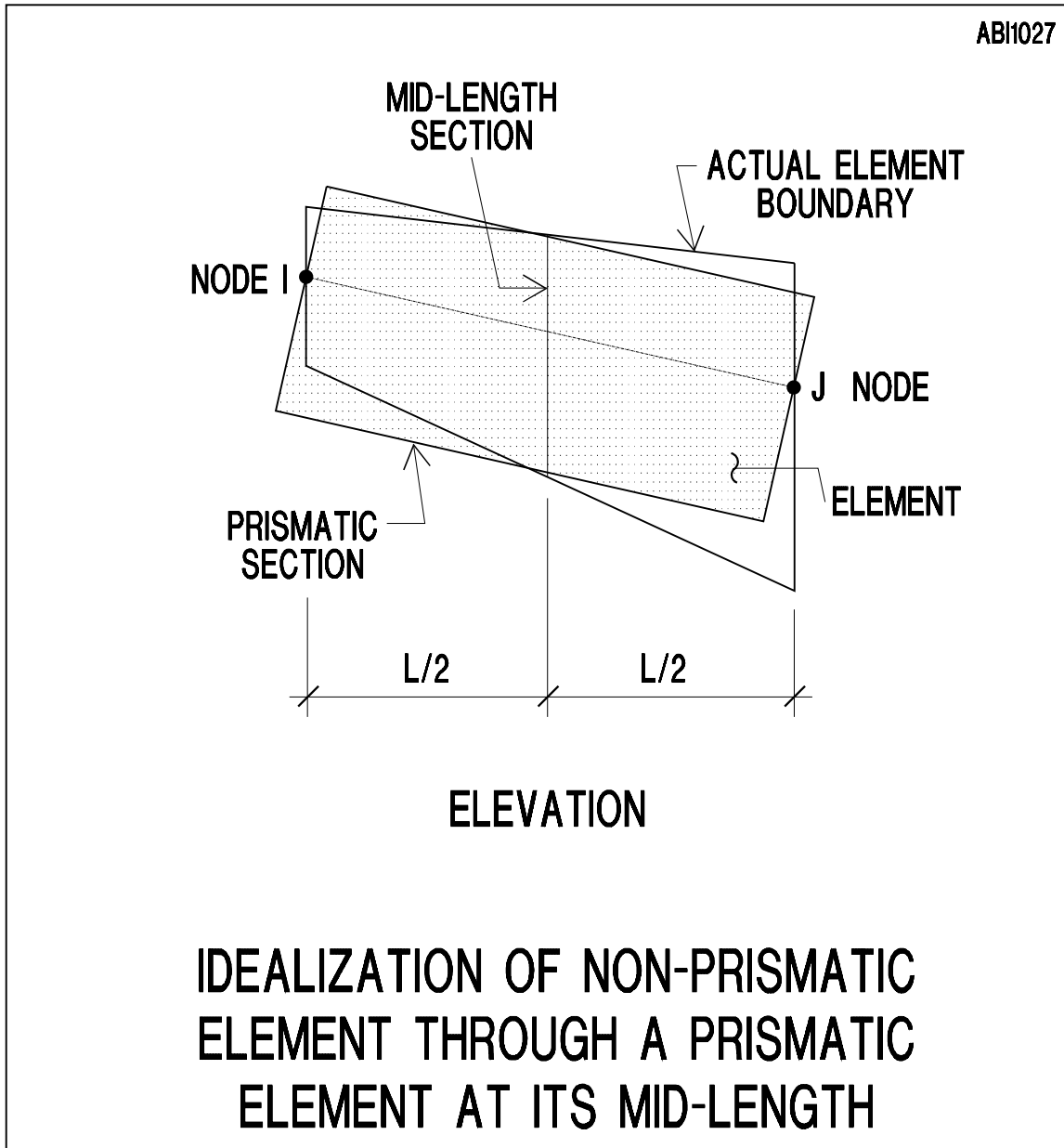


FIGURE 3.5-8

Concentrated nodal loads:

The following data must be provided. Nodal loads may be applied to any degree of freedom, whether or not it is free to displace. Loads on restrained degrees of freedom are retained for future use in case the degree of freedom is ever unrestrained.

$$N=n1,n2,inc \quad F=fx,fy,fzz$$

where,

$n1$ = Node number of first node in a series of nodes with identical loading;
 $n2$ = Node number of last node in the series [$n1$];
 inc = Node number increment;
 fx = X- force increment;
 fy = Y- force increment; and,
 fzz = Z- moment increment.

Imposed nodal displacements:

The following data must be provided. Displacements may only be applied to restrained (fixed) degrees of freedom. Applied displacements on unrestrained degrees of freedom are neglected.

$$N=n1, n2, inc \quad D=dx, dy, dzz$$

where,

$n1$ = Node number of first node in a series of nodes with identical applied displacements;
 $n2$ = Node number of last node in the series [$n1$];
 inc = Node number increment used to define nodes in the series;
 dx = X- displacement increment;
 dy = Y- displacement increment; and,
 dzz = Z- rotation increment.

Uniformly distributed frame element loads:

The following data must be provided. Frame element loads are in the global directions, and are specified as force per unit projected length (see **Fig. 3.5-9**). Frame element loads are converted to concentrated nodal forces for the analysis.

$$L=l1, l2, inc \quad F=fx, fy$$

where,

$l1$ = Frame element number of first element in a series of elements with identical applied loads;
 $l2$ = Element number of last element in the series;
 inc = Element number increment;
 fx = X- force per unit Y- projected length; and,
 fy = Y- force per unit X- projected length.

Temperature change loadings:

The following data must be provided. The temperature of each element may be different and may vary linearly over the depth of the element. Temperature strains are included in the frame elements generated with the L= identifier, and also in all prestressing tendon segments with the same node numbers as the generated elements. Temperature strains in the travelers are not included. The reference temperature for all elements is taken as the ambient temperature (see the SET command) on the day the element was installed.

$L=l1, l2, inc \quad T=ttop, tbot$

Where,

$l1$ = Frame element number of first element in a series of elements with identical temperatures;

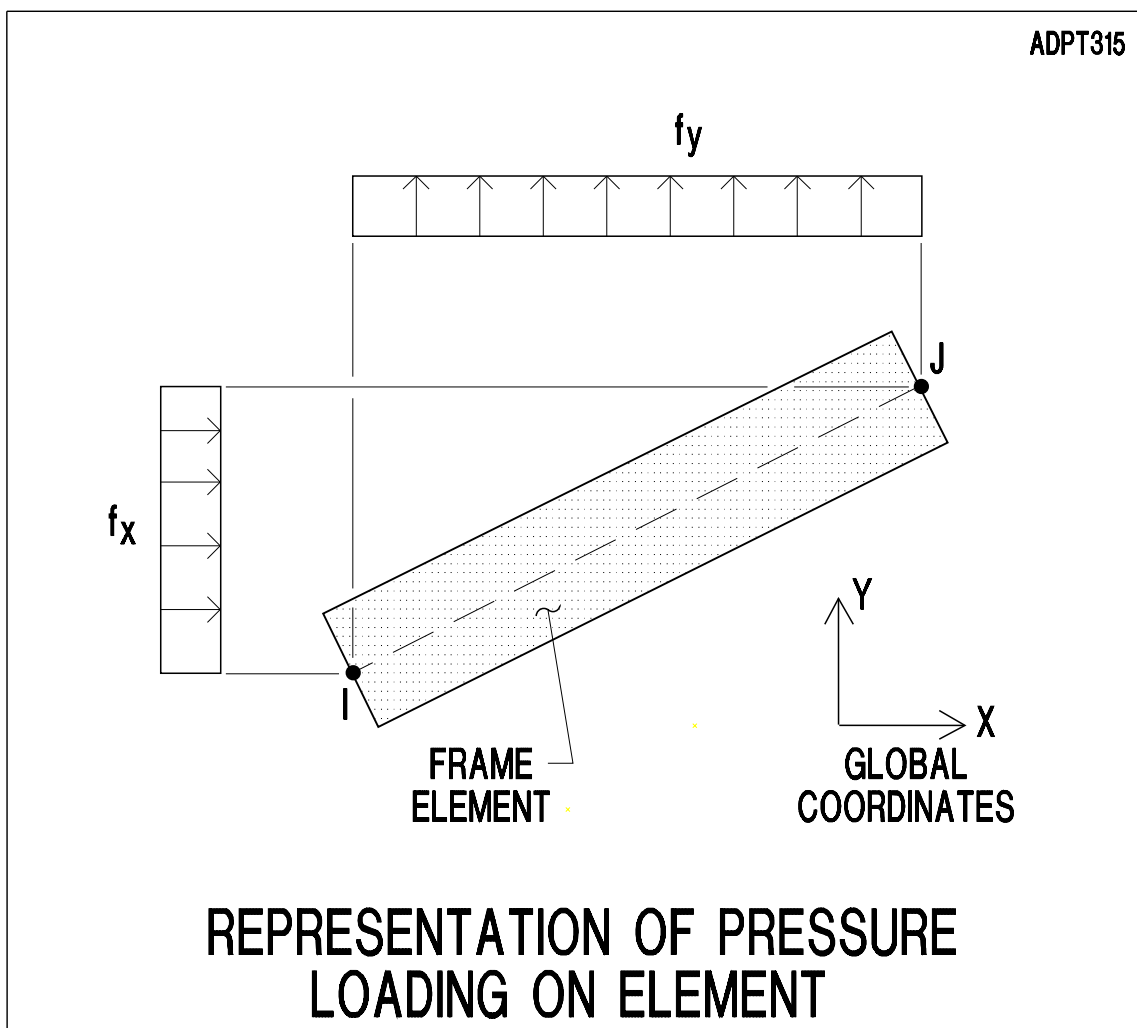


FIGURE 3.5-9

l2 = Element number of last element in the series;
inc = Element number increment;
t top = Temperature at top fiber of cross section; and,
t bot = Temperature at bottom fiber of cross section.

This sequence of lines must be terminated by a blank line.

MESH COMPLETE

Syntax: MESH COMPLETE

Explanation: The MESH COMPLETE command has no arguments. It signals the program that the mesh input phase of the analysis is complete. No mesh input commands are allowed after this command has been interpreted.

MESH INPUT

Syntax: MESH INPUT

To be followed by mesh input subcommands lines.

Explanation: The MESH INPUT command has no arguments of its own, but is followed by a series of mesh input subcommands which specify the node coordinates, material and section properties, and frame and tendon element geometries for the plane frame structure. All nodes, frame elements, tendons and travelers which will ever exist in the analysis history of the structure are defined using the mesh input subcommands. The erection sequence is specified later in the input using the RESTRAINTS, BUILD (frame element), REMOVE (frame element), STRESS (tendon) and MOVE (traveler) subcommands of the CHANGE STRUCTURE command.

The mesh input subcommands include NODES, SEQUENCE, CONCRETE PROPERTIES, MILD STEEL PROPERTIES, SECTION PROPERTIES, OFFSET DATA, FRAME ELEMENTS, PRESTRESSING STEEL, TENDON GEOMETRY, TRAVELERS, and MESH COMPLETE as described below.

The MESH INPUT command can be used **only once** in a given analysis. Also, its subcommands can be used **only once**.

MILD STEEL PROPERTIES

Syntax: MILD STEEL PROPERTIES N=?
n Es=? P=? As=?

where,

N = Total number of mild steel types [1];
n = Mild steel type number;
Es = Elastic modulus;
P = Mild steel reinforcement ratio in cross section; and,
As = Thermal expansion coefficient.

Explanation: The MILD STEEL PROPERTIES command is used to specify the different mild (passive reinforcing) steel material properties found in the structure. The mild steel type number n must be less than or equal to the total number of mild steel types input on the MILD STEEL PROPERTIES command line. Mild steel types may be supplied in any order; however each mild steel type must be specified once and only once.

Mild (reinforcing) steel is considered as a linear elastic material, uniformly distributed over the entire element cross section. The area reinforcement ratio (Steel area / Total area) defines the total amount of steel used.

A table of mild steel properties is printed in the output.

Mild steel can also be specified by its position and cross-sectional area in an element. In this case mild steel is defined as a prestressing tendons with zero initial stress. Obviously for mild steel the tendon properties shall be adjusted to reflect that of non-prestressed reinforcement.

MOVE

Syntax: MOVE N=? D=n(1) . . . n(n)

where,

N = Traveler number;
D = Destination of traveler is the location where the traveler is being installed. The traveler's position is described by the node numbers of the structure frame. Number of nodes entered herein must equal to the number of nodes specified for each traveler in TRAVELER command;
n(?) = Destination nodes for the new traveler location; and,
(n) = Number of nodes in traveler.

Explanation: The MOVE command is used to install, move and remove traveling formwork. The traveler description must have been input under the TRAVELER subcommand of the MESH INPUT command. A traveler may be moved to any

location on the structure as many times as desired during the analysis. The traveler may be removed entirely by specifying destination node $n(1) = 0$.

The dead load and stiffness of the traveler are automatically included in the analysis. When the traveler is moved to a new location, the element geometry of the traveler is adjusted to the node geometry at the new location and all loads are automatically moved.

Example: MOVE N=2 D=14, 15, 16

Means move the traveler number 2 to a new location defined by node numbers 14, 15 and 16. Note that this traveler has three nodes only. Others may have more nodes.

NODES

Syntax: NODES N=?
n X=? Y=? Scale=? G=n1, n2, inc

where,

N = Total number of nodes in the structure;
n = Node number;
X = Global X-coordinate of node n;
Y = Global Y-coordinate of node n;
Scale = Scale factor for global X- and Y- coordinates; and,
n1, n2, inc = Generation parameters described below.

Explanation: The NODES command states the total number of nodes of the structure. The node number n must be less than or equal to the total number of nodes input on the NODES command line. Node coordinates may be input or generated in any order with any number of data lines. If a node is input or generated more than once, the last specification is used.

Node coordinates are input in the global X- Y- coordinate system (**Fig. 3.5-10**). The input X- and Y- node coordinates are multiplied by the scale factor immediately after input. Once the scale factor is entered (usually on the first line of node input data) do not enter it again unless it is necessary to reset its value.

Additional node numbers and coordinates may be automatically generated using the G=n1, n2, inc parameters. Nodes and node

coordinates are generated at equal intervals along a straight line between two previously specified nodes. The node generation parameters are defined as:

$n1$ = a previously specified node number;
 $n2$ = a previously specified node number; and,
 inc = node number increment defining the generated nodes. If this parameter is left out a default of 1 is used, shown as [1].

If node coordinates are defined more than once, only the last definition will be used. The final set of coordinates for all nodes is printed in the output.

This sequence of lines must be terminated by a blank line.

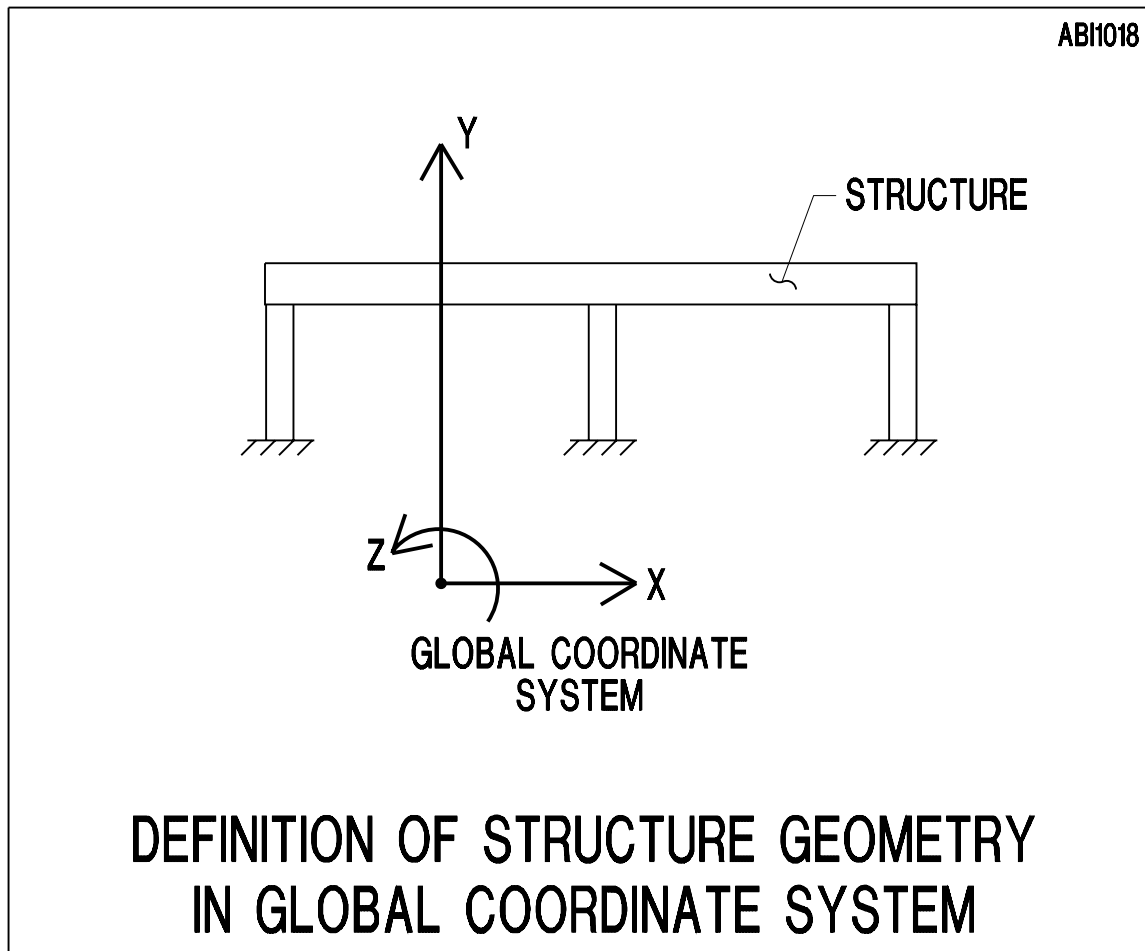


FIGURE 3.5-10

OFFSET DATA

Syntax: OFFSET DATA N=?
 n OI=?,? OJ=?,?

where,

N = Total number of different offsets for all the frame elements;
 N = Total number of offset data entries;
 n = Offset number ($n \leq N$);
 OI = x and y offset at node I (see **Fig. 3.5-6**); and,
 OJ = x and y offset at node J.

Explanation: The `OFFSET DATA` command is used to define all the possible offsets that may be used for various elements in the structure. The offset number n must be less than or equal to the total number of offset data entries input on the `OFFSET DATA` command line. The offset entries must be supplied in ascending order, starting at 1. One offset entry can only be specified once and can not be redefined.

Example:

```

OFFSET DATA N=2
  1 OI= 0.0, -200.0  OJ= 0.0, -200.0
  2 OI= 0.0, 300.0  OJ= 0.0, 300.0
  
```

OUTPUT

Syntax: `OUTPUT [NONE ALL SAME DISPLACEMENTS \`
`ACTIONS REACTIONS STRESSES, [PRESTRESSING] \`
`[PRESTRESSING LONG] [PRESTRESSING SHORT] \`
`STATIC]`

No argument = Prints all data blocks; and,
 ALL = Same as *no argument*, prints all data block.

All the other optional parameters cause data blocks 100, 101, 102, 103, 104, and 111 to be printed out together with the block specifically called by the parameter. Details of each data table is given in Appendix A.

where,

DISPLACEMENTS	= Nodal boundary conditions and nodal displacements (Block 105);
ACTIONS	= Moments, shears and axial loading on each frame element (Block 107);
REACTIONS	= Total reactions at each fixed node (Block 106);
STRESSES	= Concrete fiber stresses at points 1,2,3 and 4 (Fig. 3.5-11), (Data Block 108 in tabulated output);
STATIC	= Prints a summary of total internal forces for each frame element, summed from the frame element components plus all tendon segments and traveler elements with the same nodes;

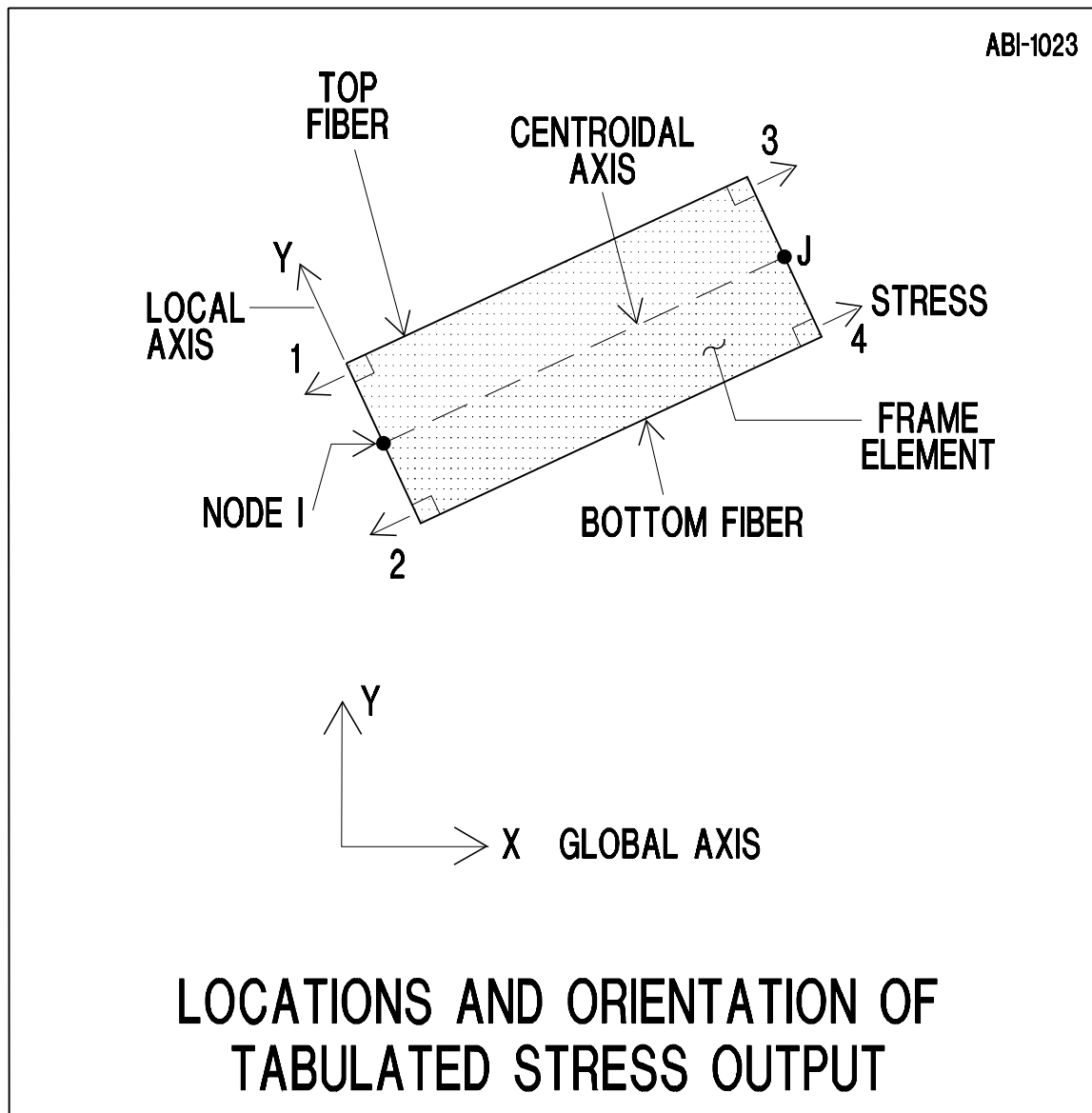


FIGURE 3.5-11

NONE	= No output for text printout will be generated. Output will be limited to graphical display;
SAME	= Output information will be the same as previous output request;
PRESTRESSING	= PRESTRESSING SHORT;
PRESTRESSING SHORT	= Prints the <i>combined</i> result of all tendons in the frame (Block 109.2); and,
PRESTRESSING LONG	= Prints a detailed output for each tendon (Block 109.1), and a summary of the combined result of all tendons (Block 109.2).

Explanation: The amount and compilation of the printed output is controlled through the OUTPUT command. Depending on the optional parameters which can follow OUTPUT command, a printout with a lesser or greater detail can be obtained.

The output is subdivided into data blocks, each numbered with a unique identification number. The numbering and details of the output data blocks are given in Appendix A.

The OUTPUT command performs no numerical operations associated with the time dependent analysis; it only prints the results.

PRESTRESSING STEEL

Syntax: PRESTRESSING STEEL N=?
 n Ep=? Meu=? K=? Fpu=? Fpy=? [R=?] Ap=?

where,

N = Total number of prestressing steel types [1];
 n = Prestressing steel type number;
 Ep = Elastic modulus;
 Meu = Angular friction coefficient (μ);
 K = Wobble friction coefficient (K);
 Fpu = Ultimate stress;
 Fpy = 0.1% yield stress for the calculation of prestress loss [0.9Fpu];
 R = Relaxation coefficient; same as “c” in relaxation equation, (see appendices) generally specified as 10 for stress-relieved and 45 for low relaxation strands; to suppress relaxation altogether, specify R=0 [0]; and,
 Ap = Thermal expansion coefficient [0]; use per Centigrade degree for SI and MKS and per Fahrenheit for American Units.

Explanation: The PRESTRESSING STEEL command is used to specify the different prestressing steel material properties found in the structure. The prestressing steel type number n must be less than or equal to the total number of prestressing steel types input on the PRESTRESSING STEEL command line. The prestressing steel types may be supplied in any order; however each prestressing steel type must be specified once.

Prestressing steel is considered as a relaxing linear elastic material. The yield stress is used only in the relaxation modeling. The friction coefficients are used in computing the initial tendon forces under the STRESS subcommand of the CHANGE STRUCTURE command. The strength is used in calculating the initial stress in tendon when the jacking is expressed as a ratio of ultimate stress. Also if Fpy is not entered, Fpu is used to estimate Fpy.

REMOVE

Syntax: REMOVE N=n1, n2, inc

where,

n1 = Element number of first frame element in a series of elements to be removed from the structure;
 n2 = Element number of last element in the series [n1]; and,
 inc = Element number increment.

Explanation: The REMOVE command is used to remove existing frame elements from the structure. The elements must have been installed using the BUILD subcommand described above. Once an element has been removed with this command, it is permanently gone from the structure and may never be installed again.

The program automatically removes the stiffness, dead load and internal forces from the system matrices when an element is removed. Any additional loads applied to the element under the LOADING command described below are **not** automatically removed, and must be removed manually (by applying an equal but opposite force with another LOADING command) **before** removing the element.

Example: REMOVE N=6
 REMOVE 5, 9, 2

In this example the frame elements 5,6,7 and 9 will be eliminated from the current configuration of the structure.

RESTRAINTS

Syntax: RESTRAINTS
 n1, n2, inc R=rx, ry, rzz M=m1, m2, minc

where,

n1 = Node number of first node in a series of nodes with identical restraint specifications;
 n2 = Node number of last node in the series [n1];
 inc = Node number increment used to define nodes in the series;
 rx = X- displacement restraint specification;
 ry = Y- displacement restraint specification;
 rzz = Z- rotation restraint specification;
 m1 = Node number of first node in a series of nodes which is master node for n1 node;

m2 = Node number of last node in the series which is master node corresponding to n2 [m1]; and,
minc = Node number increment used to define master nodes for nodes n1 through n2.

Explanation: The RESTRAINTS command is used to specify the boundary conditions of the structure. The data line is repeated as many times as required to specify the desired boundary condition changes.

Each node has three displacement degrees of freedom (**Fig. 3.5-12**), each of which may be specified with one of three restraint types:

r? = 0: Free to displace;
r? = 1: Fixed at current total displacement (current position);
r? = 2: Fixed with zero displacement; and,
r? = 3: Fixed with displacement corresponding to that of its master node. If this option is used, a master node must be defined on the same line corresponding to the node for which restraints are being defined.

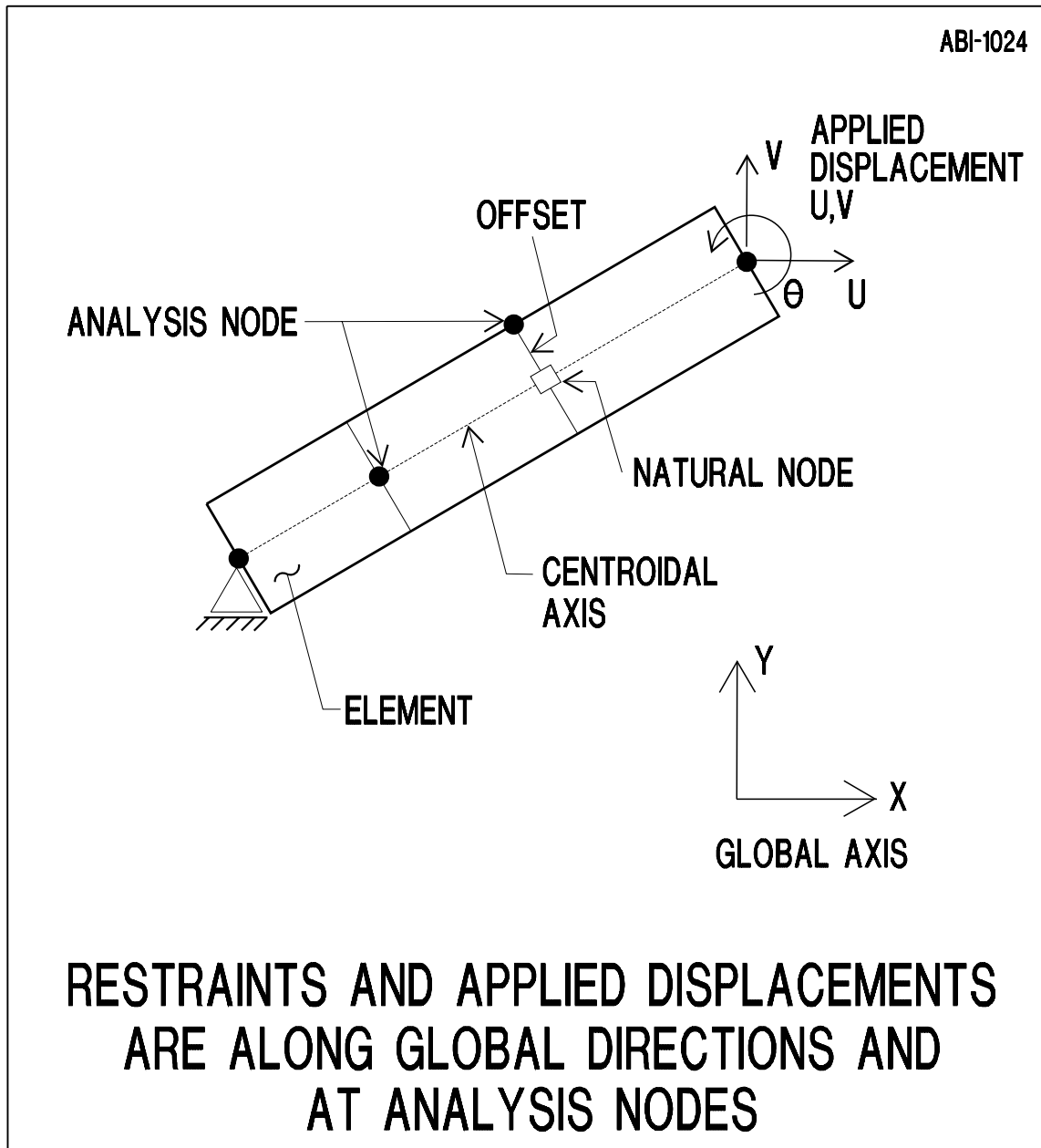
If the restraint specification is “0” the degree of freedom is unrestrained. The associated displacement and/or rotation will be computed by the program.

If the restraint specification is “1” the total displacement or rotation is restrained to its current value.

If the restraint specification is “2” the total displacement or rotation is restrained to zero. Restraint types 1 and 2 are identical except that for type 2, an imposed displacement equal and opposite to the total current displacement is applied in the next solution step.

If the restraint specification is “3”, then the associated displacement is determined from that of the master node (defined as M=?) as if the connection of the slave to the master were rigid, unless specific releases are defined as input. This master/slave relation can be present for any of the three degrees of freedom for the slave node. For a node with complete dependency, R=3,3,3 may be used. Any node defined as slave can not be set as master for another node. Also note that for two specific relationships, i.e. R=3,3,0 (a hinge condition) and R=3,0,0 the master as well as the slave nodes must have the same coordinates. The two must coincide.

Any restraint change remains in effect until changed again by a subsequent RESTRAINTS subcommand. All unspecified nodes are assumed to have a “0” (free to displace) boundary condition in all three degrees of freedom. Master / slave relation may be present in one stage but deleted in a later stage. To achieve this redefine the restraints for the node in subsequent stage to be either

**FIGURE 3.5-12**

0 or 1 or 2, without any master node definition.

The program automatically determines which nodes are defined in the current structure and includes only their unrestrained degrees of freedom in the global equilibrium equations. Thus the user need not restrain unused nodes with this command.

This sequence of lines must be terminated by a blank line.

Example:

```

12  R=1,1,1          ;Node 12 is fully clamped (fixed) in
12  R=0,0,0          ;Constraints of node 12 are removed

RESTRAINTS
  4  R=3,3,3 M=1      ;Node 4 is fully dependent
    . . . .          ;on deformations of node 1
  6  R=3,3,0 M=2      ;This models a hinge but nodes
                      ;2 and 6 must have same
                      ;coordinates

```

SECTION PROPERTIES

Syntax: SECTION PROPERTIES N=?
 n D=d1,d2,d3 B=b1,b2,b3 Area=? I=? \
 C=Ctop,Cbot S=Stop,Sbot

where,

N = Total number of section types [1]; and,
 n = Section type number.

and

d1, d2, d3 = Height dimensions of the section (**Fig. 3.5-13**); and,
 b1, b2, b3 = Width dimensions of the section (**Fig. 3.5-13**).

or

Area = Cross section area;
 I = Cross section moment of inertia;
 Ctop = Distance from centroidal axis to extreme fiber in **positive** local y-
 direction (see **Fig. 2.2-2**); and,
 Cbot = Distance from centroidal axis to extreme fiber in **negative** local y-
 direction (see **Fig. 2.2-2**).

and,

Stop = Fraction of shrinkage strain at extreme fiber in **positive** local y-
 direction [1]; and,
 Sbot = Fraction of shrinkage strain at extreme fiber in **negative** local y-
 direction [1].

Explanation: The SECTION PROPERTIES command is used to specify the different section properties found in the frame elements and travelers making up the structure. The section type number n must be less than or equal to the total number of section types input on the SECTION PROPERTIES command line. The section types may be supplied in any order; however each section type must be specified once.

An element's local coordinate is defined by direction of its node I to node J (x-direction). Node I is the first node entered in input data. Depending on the order of node entry of an element in input data, the local y-axis may look one-way, or the opposite way, since the right hand rule for coordinate definition is used. The consistent use of right-hand rule for coordinate definition may necessitate the same section to be given two different section properties, since Ctop in one may have Cbot designation in the other.

Input either the height and width dimensions (**Fig. 3.5-12**) with the D= and B= identifiers, or the section properties with the A= I= and C= identifiers. When the D= and B= input form is used, the section properties are computed by the program. When the D= and B= identifiers are entered on an input line, any additional A= I= and C= identifiers and their data will override the internally computed values.

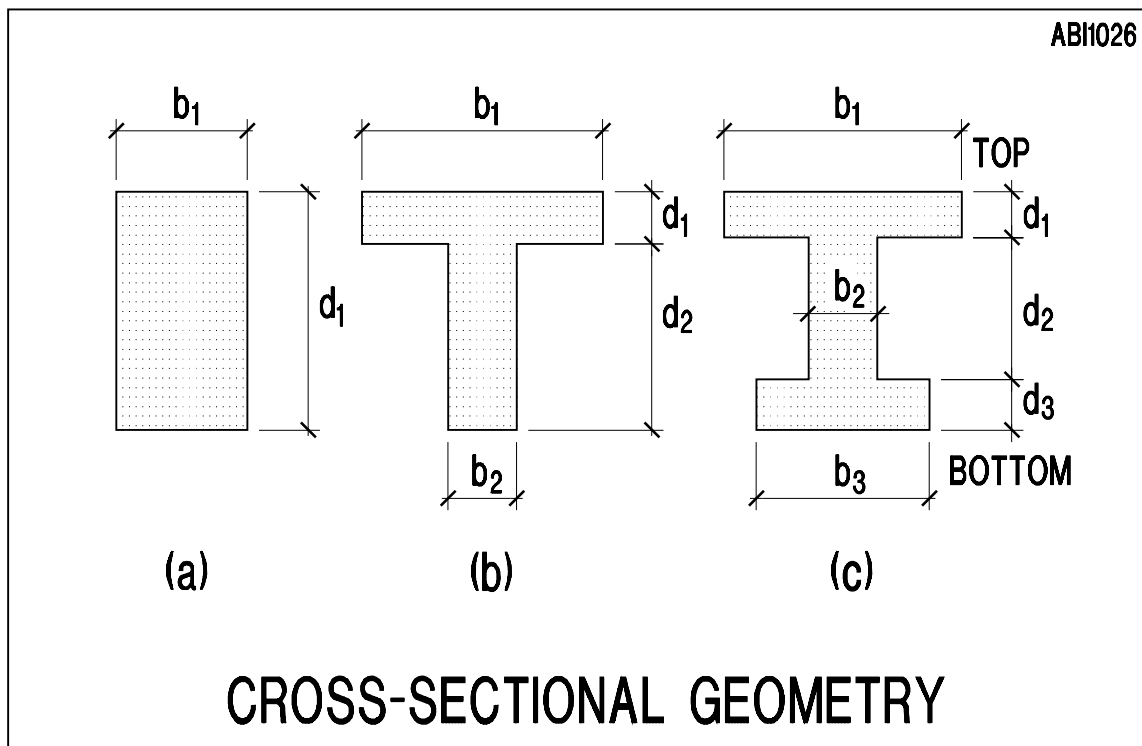
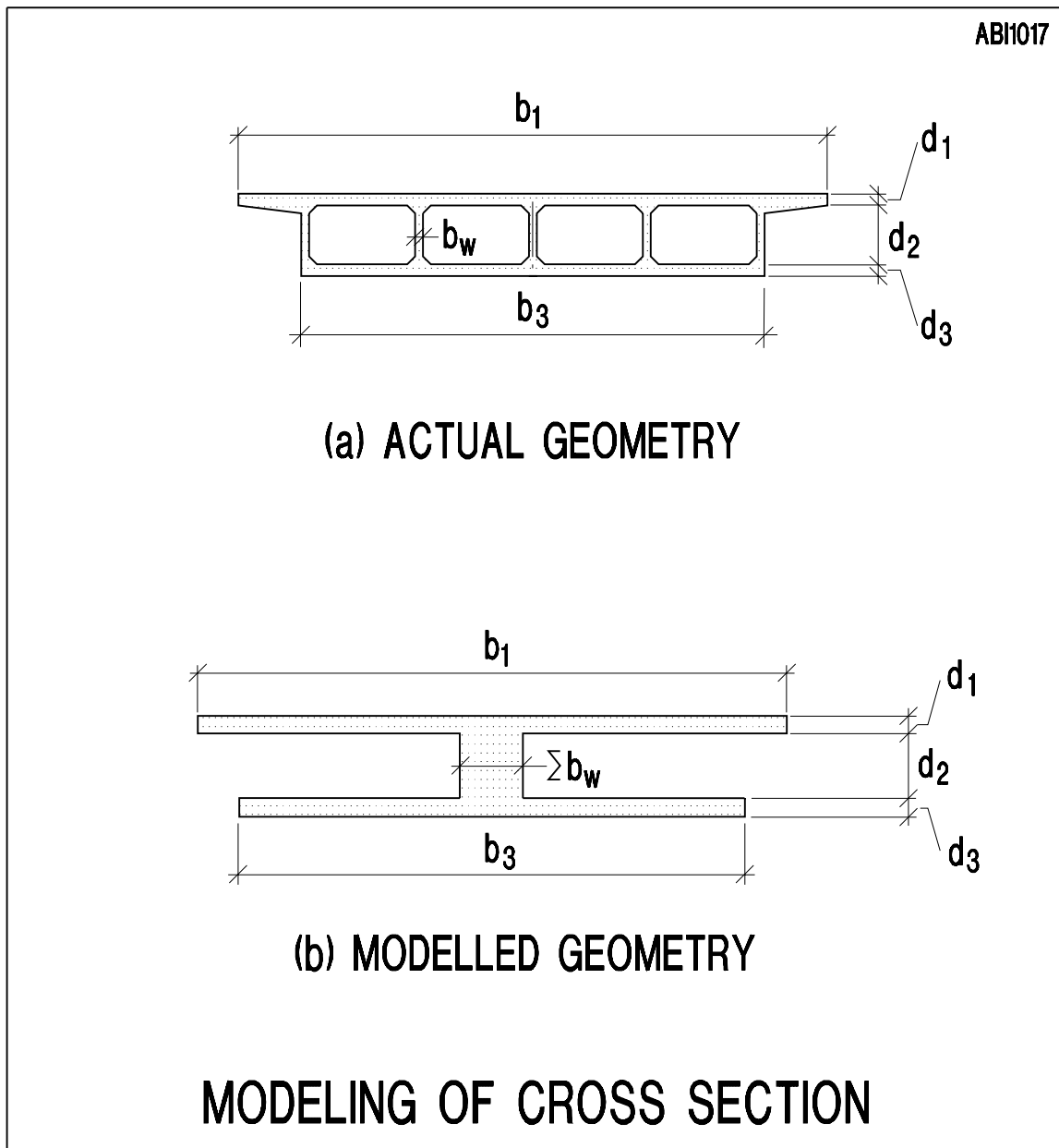


FIGURE 3.5-13

**FIGURE 3.5-14**

Figs. 3.5-14 and 3.5-15 illustrate two examples of cross-sectional idealizations used in the preparation of input data.

The S= identifier is used to model non-uniform shrinkage over the depth of the cross section. The default is uniform shrinkage.

SEQUENCE

Syntax: SEQUENCE
n1, n2, inc

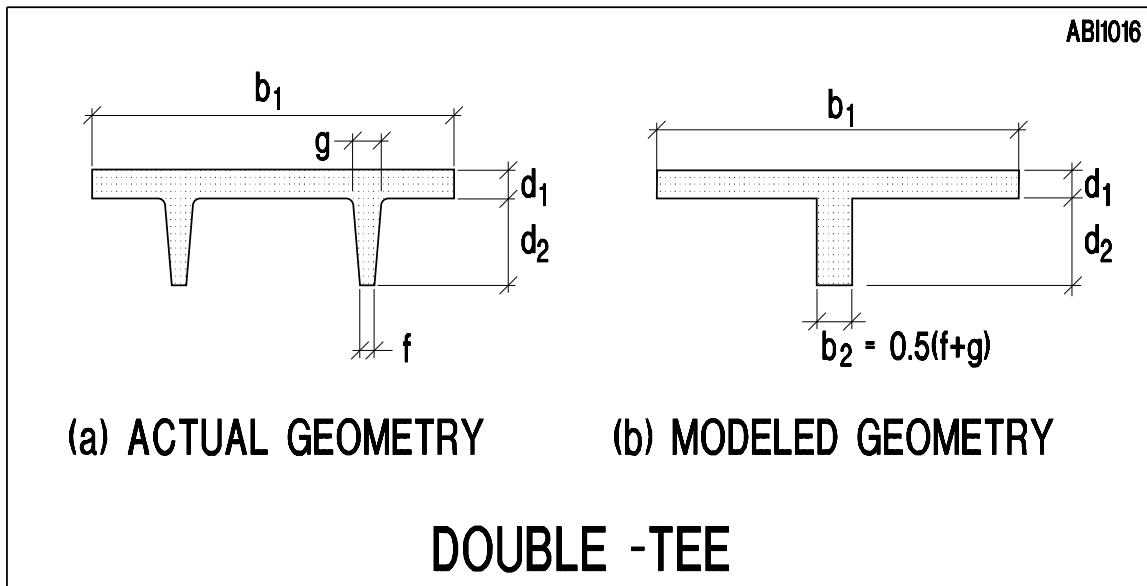


FIGURE 3.5-15

where,

- n1 = First node in a generation sequence;
- n2 = Last node in a generation sequence; and,
- inc = Node number increment [1].

Explanation: The SEQUENCE command is an optional command used to specify the node number order used for numbering the nodal displacement degrees of freedom of the structure. Skillful use of this command can reduce the profile of the stiffness matrix and increase the program's efficiency in solving the equilibrium equations. If this command is not used, degrees of freedom will be numbered in node numerical order 1

The SEQUENCE command must be placed after the NODES command and before MESH command.

Data lines must be provided to generate a list of node numbers in the order in which their degrees of freedom are to be numbered. Each node number must appear not more than once in this list. Any nodes which are omitted from this list will be numbered in numerical order after all other nodes.

This sequence of lines must be terminated by a blank line.

Example: SEQUENCE
 G=5, 9, 2
 G=1, 4

The above example is interpreted as 5, 7, 9, 1, 2, 3, 4.

SET

Syntax: SET Day=? T=? G=gx, gy C=? A=? It=?

where,

Day = Current date (days) [0];
T = Current temperature (Consistent with input units) [70 if American else 20°C];
gx = X-direction gravity load multiplier (fraction) [0];
gy = Y- direction gravity load multiplier (fraction) [-1];
C = Stress change convergence tolerance [1 psi; 0.007 Mpa; 0.07 Kg/cm³];
A = Convergence acceleration factor [0.70]; and,
It = Maximum iterations per solution step [25].

Explanation: The SET command is used to set and reset the basic environmental factors influencing the solution, as well as solution convergence and acceleration which influence the accuracy of the analysis. The command may be issued any number of times in order to change these factors as required, with the exceptions noted below.

The Day= identifier is used to set the date from which the age of the frame elements and hence the impact of time dependent effects can be calculated. The command SET DAY =? can be specified only once prior to the start of the solution (prior to CHANGE STRUCTURE command). SET command can be used more than one time with other arguments such as G=. The date can also be set under the SOLVE command.

The T= identifier is used to set the element temperatures for succeeding solution steps. This temperature is used for calculation of temperature strains only. This temperature specification overrides any previously specified temperature gradients entered using the LOADING command.

The G= identifier is used to set the load multipliers for gravity loads in the global X- and Y- directions. The gravity load multipliers gx and gy, however, influence only the gravity load increment and should be set only once, prior to any construction operations performed under the CHANGE STRUCTURE command.

The C= identifier is used to set convergence tolerances for the creep series solution. The solution is considered to have converged when the maximum stress change in the concrete in any element is less than the specified value over two successive iterations.

The A= identifier is used to set the convergence acceleration factor for the creep solution. The default value has proven satisfactory in most cases.

The It= identifier is used to set the maximum number of iterations allowed in any solution step. After this number of iterations, the solution terminates. A shorter time step or higher convergence tolerances help reduce the required number of iterations.

SOLVE

Syntax: SOLVE Day=? It=? A=?

where,

Day = Day number at end of solution;
It = Maximum iterations per time step [25]; and,
A = Convergence acceleration factor [0.70].

Explanation: The SOLVE command is used to solve the current structure for its displacements and internal stresses under current loadings at the specified time. This command performs the majority of the numerical operations required in the analysis.

All command line data are optional, and if omitted they default to their values from the previous SOLVE command or the values initialized under the SET command. Once these parameters have been entered, do not enter them again unless it is necessary to reset them. The initial default values are those described for the SET command.

The SOLVE command steps the solution over the time interval from the time at the end of the previous solution to the day number specified with the Day= identifier on the command line. All loadings are assumed to be applied gradually over the length of the time step. Thus any instantaneously applied loads require a zero length time step. A zero length time step is also required whenever the structure's configuration has been changed with the CHANGE STRUCTURE command. A zero length time step is specified by omitting the Day= identifier on the command line

The A= identifier is used to set the convergence acceleration factor for the creep solution. The default value has proven satisfactory in most cases.

The It= identifier is used to set the maximum number of iterations allowed in each time step. After this number of iterations, the solution terminates. A shorter time step or higher convergence tolerances help reduce the required number of iterations.

If the time differences between two solutions is long, due to the non-linear nature of the time parameters, it may be impractical for the program to extrapolate reasonable initial values and converge to the solution. For this reason, it is recommended to subdivide long time spans into smaller time increments.

Example: SOLVE DAY=10 ! OUTPUT
SOLVE DAY=1000 ! OUTPUT

Example: SOLVE DAY=10 ! OUTPUT
SOLVE DAY=20 ! OUTPUT
SOLVE DAY=50 ! OUTPUT
SOLVE DAY=150 ! OUTPUT
SOLVE DAY=400 ! OUTPUT
SOLVE DAY=1000 ! OUTPUT

In the first example a solution for day 1000 is attempted immediately after day=10. This might be difficult. The difficulty will show itself through high numbers of iterations (15 or more) or lack of convergence. The second example overcomes this problem by breaking the time span into smaller intervals without increase in output.

SPRINGS

Syntax: SPRINGS N=?
n,ni,nj K=kx,ky,kzz G=n1, . . . ,n5

where,

N	= Total number of frame springs [1];
n	= Spring element number;
ni	= Node number for node I; (I is the node attached to the frame for spring supports);
nj	= Node number for node J; (J is the node with fixed displacement for spring supports);
kx	= Spring axial stiffness (force per unit extension along the spring);
ky	= Spring shear stiffness (force per unit extension perpendicular to the spring);
kzz	= Spring rotational stiffness (moment per unit rotation); and,
n1 - n5	= Frame element generation parameters described below.

Explanation: The SPRINGS command is used to define the spring supports of a frame. Node, I, of the spring is defined to be attached to the frame, and node J is the fixed support. A spring is assumed to be an inert material, having only extensional and rotational stiffness at its node I. The stiffness properties of the

spring are not affected by time, creep, shrinkage and temperature. The stiffness properties remain unchanged throughout the analysis.

Additional spring elements may be automatically generated using the $G=n1, \dots, n5$ parameters. Spring elements are generated by incrementing the input parameters (above) by their respective increments, which are input using the $G=$ identifier. The generation parameters are defined as:

$n1$ = First spring in generation sequence;
 $n2$ = Last spring in generation sequence;
 $n3$ = Spring number increment;
 $n4$ = Node I increment; and,
 $n5$ = Node J increment.

Note: Springs generated using the, $G=$, generation command have all the same stiffness values.

START

Syntax: START R=?

where,

R = Data restore flag (=1 to restore database) [0]

Explanation: The START command specifies the location in the input file at which execution begins. All input lines before the START command are ignored. All input lines after the START command are interpreted as input data. Therefore, the START command may be used once in the input.

Under most circumstances, this command is the first command in the input file. It may be located later in the input file when the analysis is a restart of a prior analysis terminated with the STOP command in which case the START command must be placed subsequent to the STOP command at which the previous run has successfully terminated. For a restart case, $R=1$ should be specified on the command line so the database will be restored.

Obviously, prior to restart,

- (i) the program must have successfully executed the problem up to the command from which a re-start shall begin, and
- (ii) the data base of the former run must have been generated and be available in the subdirectory.

These steps are achieved by first placing a **STOP** command after the last **SOLVE** command of the first run. Then, after completion of the run, substitute the **STOP** by **START R=1**.

Example:

First Run	Second Run
START	-----
-----	-----
data	data
-----	-----
STOP	START R=1
-----	-----
data	data
-----	-----
-----	STOP

In the first run, the program will execute the first part of data appearing between the **START** and the **STOP**. In the second run, when encountering **R=1** after the command **START**, the program will re-create the data up to the last instruction of the first run, then it will execute the remainder of the data until the next **STOP** command.

STAY ANALYSIS

Syntax: STAY ANALYSIS Sag=[include][exclude]

where,

Sag = sag inclusion option [exclude]

Explanation: The **STAY ANALYSIS** command is used to control the type of analysis to be performed. The cable stay element formulation allows for nonlinearity in stiffness of the stay due to its sag under its weight. To include the sag correction first the following commands must be included within the **MESH INPUT** data block.

STAY ANALYSIS Sag=include

Second, the parameters of **stay** and **weight =** must be added on the tendon geometry input [refer to tendon geometry].

STOP

Syntax: STOP

Explanation: The STOP command saves the entire structure database and terminates program execution. The analysis can be restarted to analyze for more time steps by providing the saved database files and an appropriate input file for the additional steps. See the description of the START command.

STRESS

Syntax: STRESS N=n1,n2,inc Ratio=ra,rb StressTo=sa,sb \
Force=fa,fb Anchor=da,db

where,

n1 = Tendon number of first tendon in a series of tendons with identical stressing specifications;
 n2 = Tendon number of last tendon in the series [n1];
 inc = Tendon number increment;
 ra = Jacking stress ratio at tendon end "A";
 rb = Jacking stress ratio at tendon end "B";
 sa = Jacking stress at tendon end "A";
 sb = Jacking stress at tendon end "B";
 fa = Jacking force at tendon end "A";
 fb = Jacking force at tendon end "B";
 da = Anchorage slip (draw-in) at tendon end "A"; and,
 db = Anchorage slip (draw-in) at tendon end "B".

Explanation: End A of a tendon is the end first entered in the definition of tendon geometry in input data. End B is the last point of the same tendon in input data.

Example:

```
TENDON GEOMETRY  N=1
  1  N=4
    List=3,4,5,6
      1  R=5    S=7    ;End A
      2  R=7    S=8
      3  R=9    S=9
      4  R=11   S=10   ;End B
```

By reversing the order of entry the determination of A and B can be reversed.

Example: List=6,5,4,3

In this example, tendon end associated with node 6 is end A.

The STRESS command is used to install, stress, restress and remove prestressing tendons. The tendons' geometry and material properties must have been input under the TENDONS subcommand of the MESH INPUT command. Removal of tendons can be more readily achieved by using DE-STRESS subcommand.

Not all the parameters listed after the *stress* command need be specified. Use as many arguments necessary to uniquely define the intended stress condition. For example if jacking stress ratio at end A, (ra) is specified, the magnitude of stress at this point (sa) can not be specified.

A tendon is initially stressed by specifying its jacking force and anchorage slip values under this command. The tendon segment initial forces are then calculated based on this input and the material properties of the prestressing steel. A tendon may be restressed by specifying a new jacking force under a subsequent application of this command.

The jacking force is specified using either the Ratio=, or StressTo=, or Force= identifier. When the Ratio= identifier is used, the jacking force is computed using an expression of the form:

$$\text{Jacking force} = \text{Ratio} * \text{Area} * F_{pu}$$

When the Stress= identifier is used, the jacking force is computed using an expression of the form:

$$\text{Jacking Force} = \text{StressTo} * \text{Area}$$

When the Force= identifier is used, the jacking force is directly the input value.

In the first solution step after a STRESS command has been applied to a tendon, only the tendon's equivalent forces are included in the analysis. In subsequent solution steps the stiffness is also included. After the stressing operation a tendon is assumed bonded to the concrete at each nodal points. For a refined representation of unbonded tendons, the tendon need be modeled within a sheathing. This type of modeling does not impact the results of the analysis to any significant degree. For this reason in the general use both bonded and unbonded systems are modeled in the same fashion. Examples of modeling of both systems are given in the Examples Manual.

TENDON GEOMETRY

Syntax: TENDON GEOMETRY N=? [PRINT] [PRINT LONG]

This is followed by several lines of tendon geometry specification.

where,

N = Total number of prestressing tendons [1];
PRINT = Optional, if absent a table of tendon points coordinates is printed as output; and,
PRINT LONG = Optional, if present a detailed table of tendon nodes in local and global coordinates and tendon segment geometries are added. Use of this option produces voluminous output.

Explanation: The TENDON GEOMETRY command is used to specify and generate the geometry of all the prestressing tendons.

Each tendon is modeled as a series of segments connected at tendon points (**Fig. 3.5-16**). Each tendon point is rigidly constrained to a specified node, which is usually different for each tendon point. Each tendon segment must correlate entirely within one frame element. In other words there must be a one to one correspondence between each tendon segment and an associated element frame. The global X- and Y- coordinates of the tendon points completely define the tendon's geometry. These coordinates may be input directly for each tendon point, or they may be generated using a parametric generation scheme. The input or generation can be in an alternative, translated and rotated local r- s- coordinate system specified by the user. The r- and s- coordinates are converted internally to the global X- Y- system.

For input convenience, each tendon may be viewed to consist of several contiguous lengths referred to as spans. The number and lengths of tendon spans need not correlate with that of the structure. These spans are defined by the node numbers corresponding to the tendon points along their length, and need not correspond to the actual spans of the bridge. The tendon geometry of each span may be either directly input or parametrically generated.

Tables of tendon points and segment geometry may be printed in the output. The PRINT identifier on the TENDON GEOMETRY command line is used to set the amount of output and the number of tables printed.

Several lines of input, described below, are required to specify the geometry of each span of each tendon. This set of input lines must be provided for each tendon.

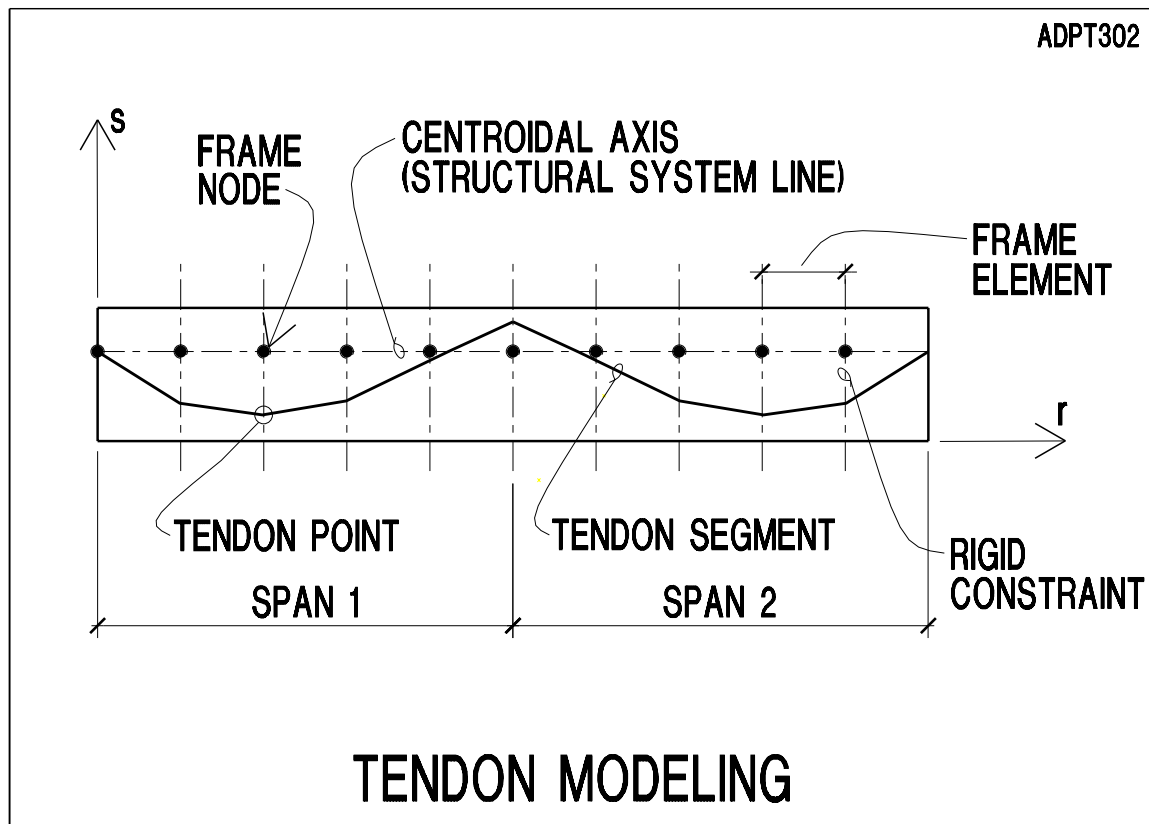


FIGURE 3.5-16

Tendon Geometry Specification:

The tendon geometry specification is repeated for each tendon. For each tendon, several input lines are made. Details of the tendon specification input lines are described next.

(i) Tendon Control Data

The first line for each tendon is the tendon control data line. Tendon numbers must be supplied in ascending consecutive order starting with tendon 1.

n Spans=? M=? Area=? [Stay Weight =?]

where,

n = Tendon identification number;

Spans = Number of spans used to define the tendon point geometry;

M = Prestressing steel type number;

Area = Cross section area for the tendon;

Weight = Weight per unit length of stay cable [lb/in in American units; N/mm in SI units and kg/cm in MKS units; program default is zero]; and,

Stay = when present, the tendon is treated as a stay.

A tendon may be regarded as a stay, if the parameter STAY appears on its control data. A stay can be stressed, restressed or removed in the same manner that a tendon is treated. If the parameter weight= is included, the program is enabled to include the sag correction for stay weight. The inclusion of sag correction is invoked under the STAY ANALYSIS command.

Example:

```
3   Spans=4   M=2   Area=4737.4
```

It means that the tendon is regarded to have been made up of four contiguous parts, each considered as a *tendon span*; it has the material property type 2 as specified in the PRESTRESSING STEEL command; and its area is 4737.4 (mm²).

(ii) Tendon Span Data

For each tendon span, several lines must be provided to input or generate the *tendon point* geometry with respect to the *frame nodes* for that span. Note that tendon spans need not correlate with the spans of the structural frame. Each tendon span is a tendon length traversing one or more frame elements. Each tendon span is made up of a number of tendon point. Each tendon point is defined in relation to a frame node. Hence the number of tendon points of a tendon span are the same as the number of frame nodes necessary to define the geometry of that tendon span.

The first tendon span data line specifies the number of nodes defining the tendon span. In addition, optionally and depending on the geometry of the tendon span, two other information items for the tendon may be entered on the same input line. These are:

- (a) The base vectors for the local r- s- coordinate system (**Fig. 3.5-17**) in which the tendon geometry is input or generated. Once these base vectors have been entered for a particular span, they need not be entered again for other spans or other tendons unless it is necessary to reset them. The initial default local coordinate system is the global X- Y- coordinate system.
- (b) The frame nodes to which the tendon points of the tendon span are associated, provided the frame nodes can be generated using the, G=, automatic generation feature described in the following.

The syntax of the first line of tendon span input is:

```
ns N=? [G=n1,n2,inc] [B=x0,y0 E=x1,y1]
```

where,

ns = Span number;
 N = Number of nodes in span (including both end nodes);
 $n1$ = Node number at left end of span;
 $n2$ = Node number at *right* end of span;
 inc = Node number increment for generating nodes in the span
 $x0, y0$ = Global coordinates of the origin of the local r-s coordinate system [0,0]; and,
 $x1, y1$ = Global coordinates of a point on the local r-axis [1,0].

Span numbers, ns , must be supplied in ascending consecutive order starting with span 1.

If the node numbers cannot be generated for a particular span, then the optional $G=$ identifier above should be excluded, and the next line must provide a list of the node numbers included in the span, in order from start (typically the left point (A)) to the end (typically right point (B)) in the span. If the optional, $G=$ identifier is used, then this line (list) must not be provided.

List= $n(1), \dots, n(N)$

where,

$n(?)$ = Node number included in the span; and,
 N = Number of nodes in the span (input on previous line).

Example 1:

2 $N=7$ $G=8, 14, 1$ $B=12, 36$ $E=360, 92$

It means that tendon span 2 has 7 tendon points. These tendon points are associated with the frame nodes 8, 9, 10, 11, 12, 13, 14 . In the input lines to follow, the geometry of this tendon span is going to be defined with respect to a set of local Cartesian coordinate axes (r, s) which has its origin at the global point $X=12, Y=36$. The, r, axis passes through the global point $X=360, Y=92$.

Example 2:

3 $N=7$ $G=8$ 14 1

It conveys the same specification as in Example 1, with the difference that no local coordinate axes (r,s) for definition of tendon nodes are specified. The program uses the local coordinate system associated with the preceding tendon span. If none is defined in the preceding lines, the global coordinate system is used.

Example 3: 2 N=7
 List=8, 9, 10, 14, 26, 5, 32

It means that tendon span number 2 has 7 *tendon* points associated with the 7 *frame* nodes. The 7 tendon points 1, 2, 3, 4, 5, 6, 7 are associated with the frame nodes 8, 9, 10, 14, 26, 5, 32 respectively.

The next line(s) provide data for generation or direct input of tendon point coordinates for the span using several options.

Either:

If parametric tendon point generation is used for the span, one line with the following data must be provided:

R=r_l, r_c, r_r S=s_l, s_c, s_r

where,

r_l = Fraction of the total span length between the *left* end of the span and the *left* inflection point (a positive number) (**Fig. 3.5-17**);
 r_c = Fraction of the total span length between the *left* end of the span and the point of zero tendon slope (*low point*) relative to the r- s- system (a positive number) (**Fig. 3.5-17**);
 r_r = Fraction of the total span length between the *right* inflection point and the *right* end of the span (a positive number) (**Fig. 3.5-17**);
 s_l = s-coordinate of left end of tendon (**Fig. 3.5-17**);
 s_c = s-coordinate of point of zero slope (**Fig. 3.5-17**); and,
 s_r = s-coordinate of right end of tendon (**Fig. 3.5-17**).

Example: r=0, 0.4, 0.1 s=12, 2, 16

The values given for , r , are dimensionless, but those for , s , are ordinates in the local s-direction.

Or:

If direct tendon point coordinate input is used for the span, the following, N, lines must be provided in place of the one above. In this case the default, r-coordinates are the x-coordinates of the nodes included in the span. If parametric generation is used, then these lines must not be provided.

np R=r(np) S=s(np)

where,

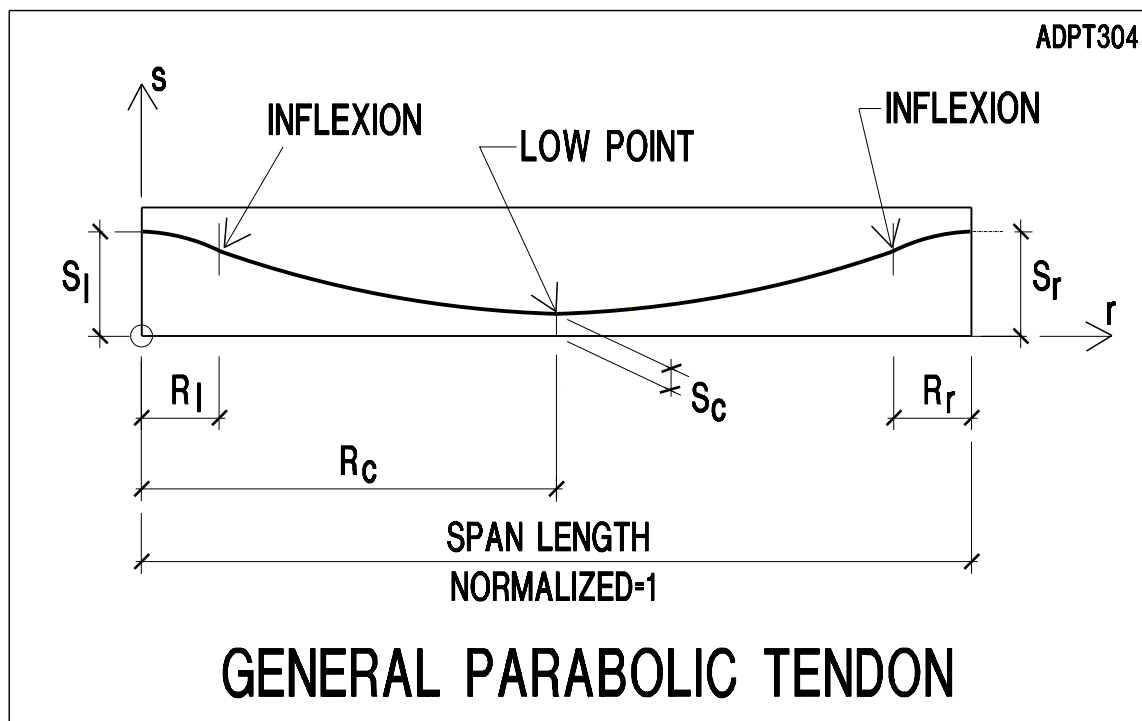


FIGURE 3.5-17

np = Tendon point number;
 $r(np)$ = r -coordinate of tendon point; and,
 $s(np)$ = s -coordinate of tendon point.

This sequence of lines must not include any blank lines.

Example:

```

2  N=7
   List=8,9,10,11,16,3,14
     1  r=0      s=0
     2  r=60     s=-5.5
     3  r=120    s=-8.9
     4  r=180    s=-10
     5  r=240    s=-8
     6  r=300    s=-2
     7  r=360    s=8
  
```

It means that tendon span number 2 has seven points. The frame nodes to which these are associated are: 8,9,10,11,16,3,14. Tendon node (point), 5, is associated with frame node 16. The coordinates of the tendon node (point) 5 are $r=240$, $s=-8$. The coordinates of the frame node 16 are defined elsewhere (in MESH INPUT command). Since the coordinates of tendon node 5, defined herein, is not likely to match those of its associated frame node, 16, the tendon point, 5, will have an x - and y -offset with respect to the frame node 16. The software computes and outputs the offsets.

TITLE

Syntax: TITLE N=? [1]
 Problem ID text line 1
 .
 Problem ID text line ?

Explanation: The TITLE command prints a program identifier in the output file, and then prints N lines of text provided on the N input lines immediately following the command line. The TITLE command is optional, but should be the first command interpreted, in order to clearly identify the output file.

TRAVELERS

Syntax: TRAVELERS N=?
 n X=? E=? W=? N=?

where,

N = In the first line (heading) total number of travelers; in the following lines, number of nodes in traveler;
 n = Traveler number;
 X = Cross section type number;
 E = Elastic modulus of material; and,
 W = Total weight of traveler (lb in American units; Newtons in SI units and kg in MKS units.

Explanation: The TRAVELERS command is used to describe all the traveling formwork used in modeling the construction sequence. Traveler numbers must be supplied in ascending consecutive order starting with traveler 1.

Travelers are modeled as linear elastic frame elements linearly connecting several nodes. Only the traveler properties are input under this command. The locations of the travelers may change at any time and are input under the MOVE subcommand of the CHANGE STRUCTURE command.

UNITS

Syntax: UNITS U=?

where, the following three systems of units are available

U=SI
 or,
 U=USA

or,

U=MKS

SI = All distances in, mm (millimeter); all forces in, N (Newton);

USA = All distances in, in. (inch); all forces in, lb (pound); and,

MKS = All distances in, cm (centimeters); all forces in, kg (Kilograms).

This Page Left Intentionally

BLANK

4. PROGRAM BACKGROUND

4.1 FRAME ELEMENT FORMULATION	3
4.1.1 Assumptions and Scope.....	3
4.1.2 Geometry of Internal Displacement Fields	3
4.2 INTERNAL DEGREES OF FREEDOM	7
4.3 MATERIAL CONSTITUTIVE RELATIONSHIPS	8
4.3.1 General Assumptions.....	8
4.3.2 Concrete Component	8
4.3.3 Nonprestressed Steel Component.....	11
4.4 PRESTRESSING TENDON FORMULATION	12
4.4.1 Tendon Discretization.....	12
4.4.2 Tendon Geometry Definition.....	12
4.4.3 Determination of Initial Tendon Forces.....	17
A. Friction losses.....	17
B. Anchor slip (seating) losses.....	19
C. Influence of stressing procedure.....	20
D. Discretized segment forces.....	20
E. Equivalent prestressing loads.....	22
4.4.4 Material Constitutive Relationships.....	22
4.4.5 Stiffness and Load Computation	26
A. Segment stiffness matrix	27
B. Segment internal force due to nodal displacements.....	27
C. Equivalent nodal loads due to initial forces.....	28
4.5 TRAVELING FORMWORK FORMULATION	28
4.5.1 Traveler Function and Purpose.....	28
4.5.2 Traveler Geometry.....	29
4.5.3 Material Constitutive Relationship	29
4.5.4 Stiffness and Load Computation	30

This Page Left Intentionally

BLANK

This Chapter presents an overview of the theoretical background to the formulation of the program. Detailed description of critical formulation is given in the appendices and the references quoted.

4.1 FRAME ELEMENT FORMULATION

4.1.1 Assumptions and Scope

The plane frame element used in the formulation of the program is based on classical Bernoulli-Euler beam kinematics. Its stiffness matrix and load vector terms include the effects of axial and bending deformations only. Shear deformation is neglected, since it is generally of little or no significance for the concrete structures covered by this work. The element formulation for this element and all other elements used in the program guarantees static equilibrium of total internal forces with the total externally applied loads.

The element consists of parallel concrete and mild steel components, for modeling the typical composite concrete and steel bridge girder. Time dependent concrete strains including mechanical strain $\epsilon^m(t)$, aging strain $\epsilon^a(t)$, creep strain $\epsilon^c(t)$, shrinkage strain $\epsilon^s(t)$ and temperature strain $\epsilon^T(t)$ are automatically considered, using the formulation discussed in Appendix B. Shrinkage and temperature strains may vary linearly over the depth of each element. The mild steel component is assumed to be uniformly distributed over the entire cross section. Concrete and steel instantaneous stress - mechanical strain laws are limited to linear elasticity, focusing the applicability of the program to the service load range of fully prestressed structures.

Where stresses exceed the cracking limit of concrete, the serviceability solutions obtained are approximate. The degree of approximation depends on the level of hypothetically calculated tension stress beyond the concrete cracking limit. For the determination of sectional actions (moments, shears, axial forces) it is common to assume full effectiveness of cross section. For deflections, however, adjustments to the calculated values are necessary (PTI 1990), when hypothetically calculated tension stresses exceed $12(f_c')^{1/2}$ ($1.0 (f_c')^{1/2}$ in SI units).

4.1.2 Geometry of Internal Displacement Fields

The geometry of the frame element (**Figure 4.1-1(a)**) is defined by the two nodes I and J located in the global X- Y- plane and on the centroidal axis of the element. The origin of the local x- y- coordinate system is at node I. The local x- axis is defined by the vector joining node I and node J. The local z- axis is parallel to and in the direction of the global Z- axis. The local y- axis

is orthogonal to the local x- and z- axes and is directed according to the right hand rule.

The element cross section (**Figure 4.1-1(b)**) is defined in the local y- z- plane and must be constant over the (x-) length of the element. The only constraint on the shape of the cross section is that it must be symmetric about the local y- axis. Any arbitrary cross section which meets this requirement can be modeled.

The element has six global displacement degrees of freedom (**Figure 4.1-1(c)**), consisting of two translations and one rotation at each node. All external loads are assumed to be applied at the nodes. Any distributed loads, including dead loads, are converted to statically equivalent concentrated nodal actions. Any element local internal effects of external loads are neglected.

The internal displacement fields in the element are expressed in terms of the three independent local nodal displacements θ_i , θ_j and u_j (**Figure 4.1-2(a)**) using cubic hermitian polynomial displacement interpolation functions (Equation 4.1.1). These interpolation functions represent exactly classical Bernoulli-Euler beam kinematics and can be integrated exactly for linear elastic materials.

The axial displacements u_0 (**Figure 4.1-2(a)**) along the element's x- axis can be expressed in terms of the local nodal displacements as:

$$\{u_0\} = [\phi_1 \quad \phi_2 \quad \phi_3] \begin{Bmatrix} \theta_i \\ \theta_j \\ u_j \end{Bmatrix} \quad (4.1.1)$$

where,

$$\phi_1 = x - \frac{2x^2}{L} + \frac{x^3}{L^2} \quad (4.1.1a)$$

$$\phi_2 = \frac{x^3}{L^2} + \frac{x^2}{L} \quad (4.1.1b)$$

$$\phi_3 = \frac{x}{L} \quad (4.1.1c)$$

The axial displacement, u_x , at any point in the element can be expressed in terms of the displacements u_0 and v_0 along the element's x- axis as

$$u_x = u_0 - y \frac{dv_0}{dx} \quad (4.1.2)$$

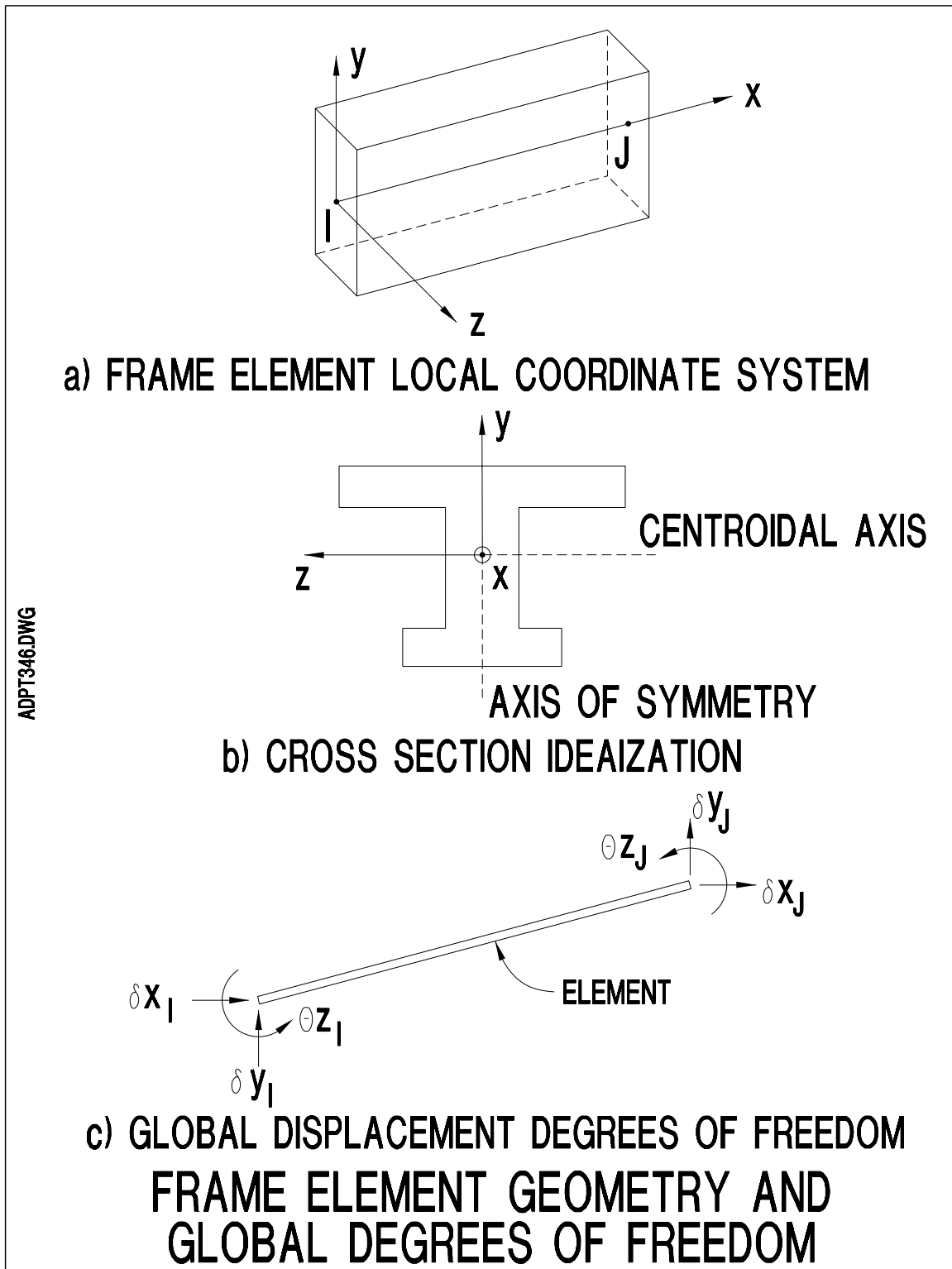


FIGURE 4.1-1

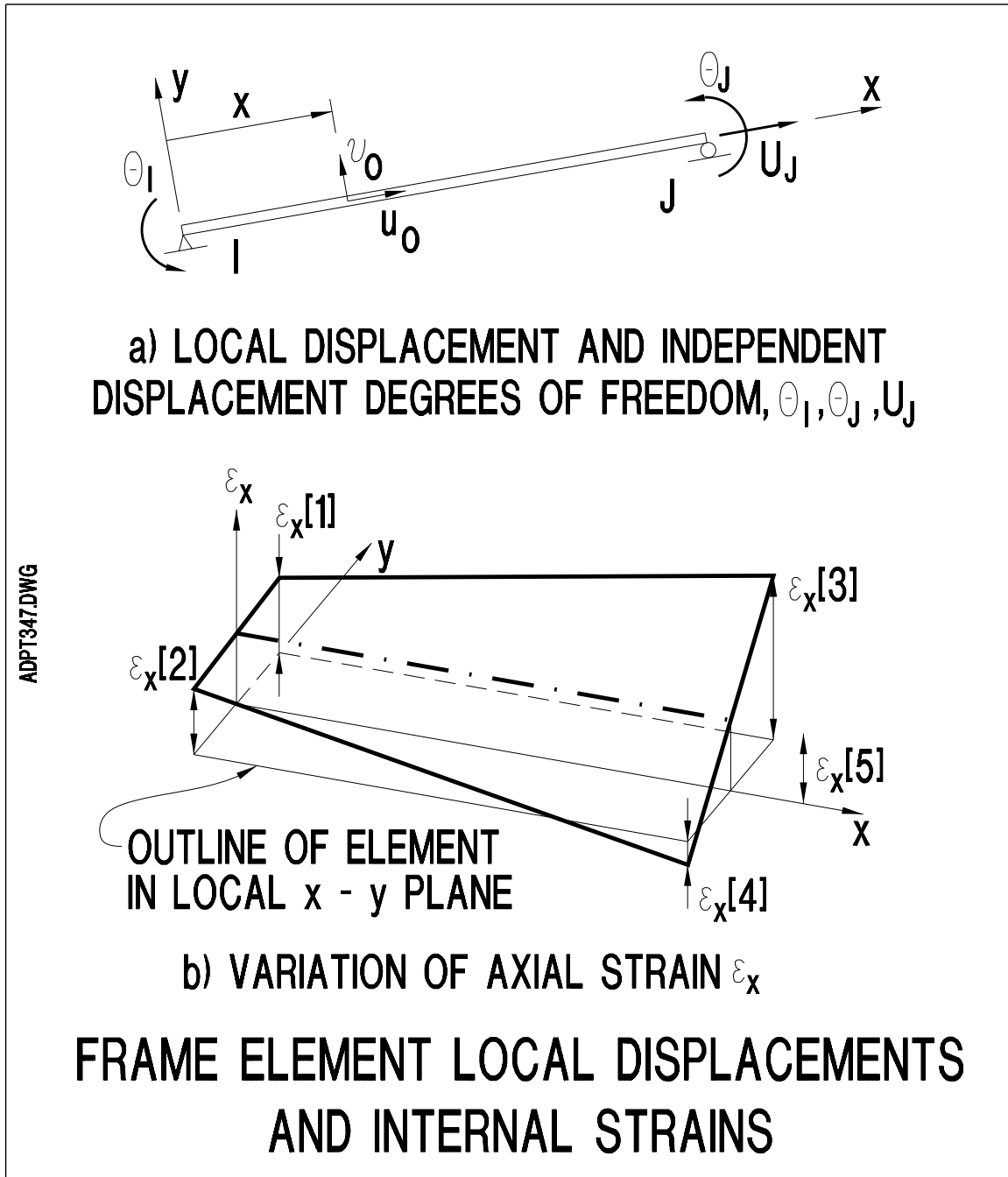


FIGURE 4.1-2

The axial strain ϵ_x is obtained by differentiating u_x with respect to x :

$$\begin{aligned}\epsilon_x &= \frac{du_x}{dx} \\ &= \frac{du_0}{dx} - y \frac{d^2 v_0}{dx^2}\end{aligned}\quad (4.1.3)$$

Substituting Equation 4.1.1 into Equation 4.1.3, the resulting strain-displacement relationship can be expressed as:

$$\epsilon_x = \left[\begin{pmatrix} y & \phi_{1,xx} \end{pmatrix} \begin{pmatrix} y & \phi_{2,xx} \end{pmatrix} \begin{pmatrix} \phi_{3,x} \end{pmatrix} \right] \begin{Bmatrix} \theta_i \\ \theta_j \\ u_j \end{Bmatrix} \quad (4.1.4)$$

where $_{,x}$ and $_{,xx}$ represent the first and second derivatives with respect to x .

4.2 INTERNAL DEGREES OF FREEDOM

Investigation of the strain-displacement relationship (Equation 4.1.4) shows that under nodal loading, axial strain ϵ_x varies linearly in both the x - and y - directions, and at the centroidal axis ($y=0$), ϵ_x is constant over the (x -) length of the element (**Figure 4.1-2(b)**). This means that the strain (or stress) distribution within the element may be described uniquely by the constant strain (or stress) at the centroidal axis plus the strain (or stress) at some other y - coordinate at two different (x -) cross sections along the element. Note that as with the traditional independent displacement degrees of freedom θ_i , θ_j and u_j (**Figure 4.1-2(a)**), three terms describe uniquely the strain (or stress) distribution within the element.

The x - and y - coordinates for tracking strains and stresses are located at the (y -) extreme fibers of the cross section and at the (x -) extreme ends of the element (**Figure 4.2-1**).

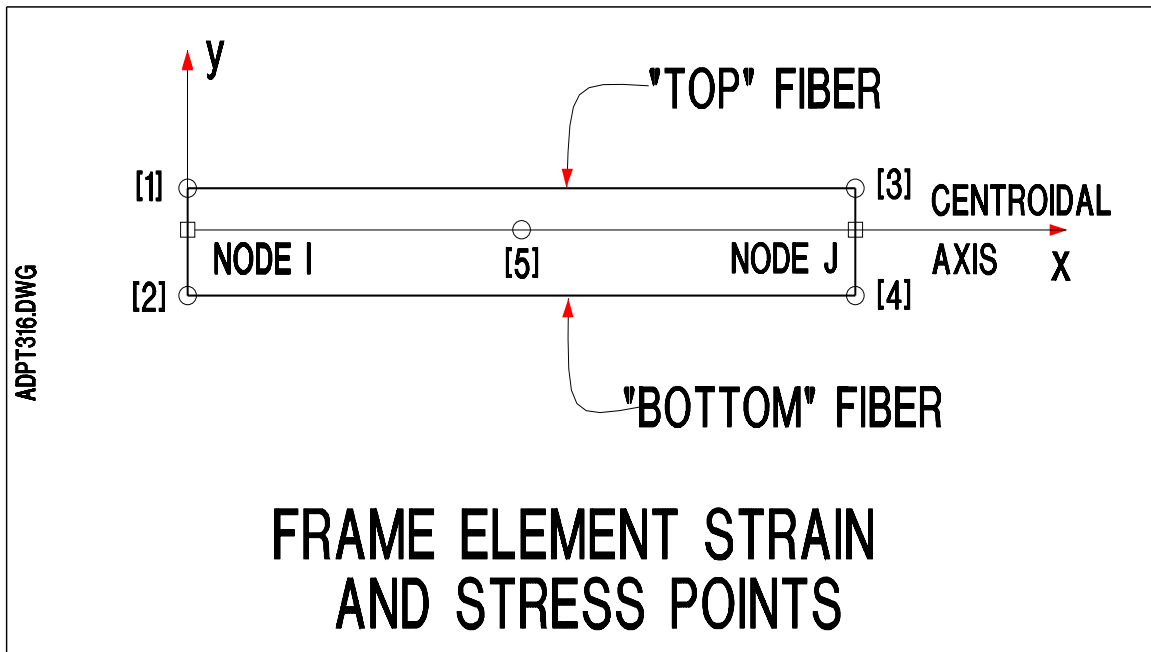


FIGURE 4.2-1

The linear variation of stresses and strains within the element simplifies the integrations required in the computation of the element characteristic matrices. All integrations over the volume of the element are performed exactly, eliminating the need for numerical integration for stiffness and load terms and reducing most such integrations to simple matrix transformations. A simple matrix transformation from stresses to stress resultants is also performed, so that traditional element-end stress resultants can be output in addition to the stresses and strains at the integration points. In the following, these integration and transformation matrices are derived and discussed in detail.

4.3 MATERIAL CONSTITUTIVE RELATIONSHIPS

4.3.1 General Assumptions

Each frame element consists of parallel concrete and mild steel components. The interaction between these components is handled on the structure level by enforcing compatibility at the nodes and therefore within the elements. Thus the constitutive relationships for the two components are uncoupled.

In Bernoulli-Euler beam theory, only the x- stress components contribute to the virtual work expressions. Thus the constitutive relationships for stresses in terms of strains are one dimensional.

4.3.2 Concrete Component

The concrete component is considered as an aging viscoelastic material using the formulation discussed in Appendix B. Refer to Appendix B for a more complete discussion of the mathematical model. The constitutive relationship is given by

$$\sigma = E [\epsilon - \epsilon''] \quad (4.3.1)$$

where,

$$\begin{aligned} \sigma &= \text{total stress at time } t_j \\ E &= \text{current elastic modulus} \\ \epsilon &= \text{current total strain} \\ \epsilon'' &= \text{current pseudo-inelastic strain} \\ \epsilon - \epsilon'' &= \text{current mechanical strain} \end{aligned}$$

The pseudo-inelastic strain ϵ'' consists of several components. To describe a time step, let time be subdivided by discrete times t_j , and let δ refer to increments from t_{j-1} to t_j . Then:

$$\begin{aligned}\delta t &= t_j - t_{j-1}, \\ \delta \epsilon^c &= \epsilon^c(t_j) - \epsilon^c(t_{j-1}), \\ &\dots\end{aligned}$$

The creep strain increment $\delta \epsilon^c$, the shrinkage strain increment $\delta \epsilon^s$ and the temperature strain increment $\delta \epsilon^T$ all contribute to the virtual work expression and contribute as initial strains to the load vector for the time step. The aging strain increment $\delta \epsilon^a$ is a fictitious strain due to the change in elastic modulus which does not contribute to the load vector but does enter the constitutive relationship. The total pseudo-inelastic strain is found by summing these component.

$$\delta \epsilon^{"}_j = \delta \epsilon^c + \delta \epsilon^s + \delta \epsilon^T + \delta \epsilon^a \quad (4.3.2)$$

$$\epsilon^{"}_j = \epsilon^{"}_{j-1} + \delta \epsilon^{"}_j \quad (4.3.3)$$

The significance of these strain components contributing to the total strain at any point is discussed in detail in Appendix B. In this chapter only the relationships required for their calculation are discussed.

The creep strain increment $\delta \epsilon^c$ is a function of the previous stress history and of the stress change during the current time step. $\delta \epsilon^c$ can be calculated recursively assuming either model,

- A. Constant stress and constant material parameters over each time step; or
- B. linear variation in stress and constant material parameters over each time step; or
- C. linear variation in stress and linear variation in material parameters over each time step.

For model a, Equation B.3.11 (Appendix B) applies for computing $\delta \epsilon^c$. For model b, Equation B.3.14 applies for computing $\delta \epsilon^c$. For model c, Equation B.3.19 is used for computing $\delta \epsilon^c$. The model used in the program is type, c, that is to say, linear variation in stress and linear variation in material properties over each time step in the analysis.

The assumed variation in stresses over a time step is fundamental to each creep model and must be considered when external loads are applied to the structure. For model a, all stress changes take place instantaneously at the ends of the time steps, therefore all loads are also assumed to be applied instantaneously at the ends of the time steps. For models, b, and, c, all

stresses vary linearly over the length of the time steps, therefore all loads are assumed to be applied gradually over the length of the time steps. Therefore, for models b and c, any loads applied instantaneously must be applied in a zero length time step.

Models b and c for the calculation of the creep strain increment require iteration during each time step, since the creep strain increment $\delta\epsilon^c$ both contributes to the incremental load vector and also depends on the stress increment over the time step. In the first iteration the stress change over the time step is taken as zero, the displacement increment is computed, the constitutive relationship is evaluated, and a first estimate of the actual stress change over the time step is computed. This computed stress change estimate is then used in the constitutive model and the process is repeated for a better estimate of the actual stress change. The iteration continues until the stress change, $\delta\sigma_x$, converges to a specified tolerance.

In some cases the iterative solution converges very slowly, therefore a simple convergence acceleration scheme has been incorporated in the program. In each iteration, the creep strain estimate is corrected using the following relationship:

$$\delta\epsilon_i^c = \frac{\delta\epsilon_i^c + C\delta\epsilon_{i-1}^c}{1 + C} \quad (4.3.4)$$

where,

- C = the convergence acceleration factor,
- $\delta\epsilon_i^c$ = corrected creep strain estimate for iteration i, and
- $\delta\epsilon_i^c$ = uncorrected creep strain estimate for iteration i

This scheme reliably accelerates convergence in cases where the computed solution oscillates around the final solution. It will slow convergence when the solution converges monotonically, however in such cases convergence is very fast and the impact of the slowed convergence on total solution time is very small. An acceleration factor of, $C = 0$, is equivalent to no acceleration. The value, $C = 0.71$, was used in the numerical examples discussed in Chapter 5 and is the default value for the program.

The shrinkage strain increment $\delta\epsilon^s$ is a function of the age of the concrete t_j and the ultimate shrinkage strain ϵ_u^s . It may vary linearly over the (y-) depth of each element and is constant over the (x-) length of the element. It may be expressed as

$$\delta\epsilon^s = \epsilon^s(t_j) - \epsilon^s(t_{j-1}) \quad (4.3.5)$$

The aging strain increment $\delta\epsilon^a$ is a fictitious strain used to take into account the increase in elastic modulus $E(t)$ over the time step. During the time step δt the elastic modulus changes from $E(t_{j-1})$ to $E(t_j)$. In the absence of creep, shrinkage and a load increment, the total stresses σ and strains ϵ remain constant. Therefore $\delta\epsilon^a$ is introduced as an increment in the pseudo-inelastic strain so that the constitutive relationship still applies:

$$\delta\epsilon^a = \sigma_x(t_{j-1}) [1/E(t_{j-1}) - 1/E(t_j)] \quad (4.3.6)$$

The temperature strain increment $\delta\epsilon^T$ over the time step δt is given by

$$\delta\epsilon^T = \delta T \alpha = [T(t_j) - T(t_{j-1})] \alpha \quad (4.3.7)$$

where $T(t)$ is the temperature T at time t and α is the coefficient of thermal expansion which is a material property. The temperature at any time may vary linearly over the (y-) depth of each element and must be constant over the (x-) length of the element.

Values of all the time dependent material parameters $E(t)$, $a_i(t)$, $\epsilon^s(t)$, etc. are found from tables generated internally by the program. Prior to the program's solution phase, these tables are initialized with the values of the material parameters at a number of different concrete ages. For concrete ages different from those tabulated, a linear interpolation between the nearest tabulated values is used. The calculation of these material parameters is discussed in Appendix B.

The integrations of the appropriate stresses and pseudo-inelastic strain components over the volume of the element to arrive at the equivalent loads representing these stresses and strains are performed by matrix transformations of the stress and pseudo-inelastic strain vectors.

4.3.3 Nonprestressed Steel Component

The nonprestressed steel component is considered as a linear elastic material. Its constitutive relationship is given by

$$\sigma = E [\epsilon - \epsilon^T] \quad (4.3.8)$$

where,

$$\begin{aligned} \sigma &= \text{total stress at time } t_j \\ E &= \text{constant elastic modulus} \\ \epsilon &= \text{total strain at time } t_j \\ \epsilon^T &= \text{temperature strain at time } t_j = [T(t_j) - T(\text{ref})]\alpha \end{aligned} \quad (4.3.9)$$

$T(t)$ is the temperature T at time t and α is the coefficient of thermal expansion. The coefficient of thermal expansion α may be different for the steel and concrete. The temperature at any time may vary linearly over the (y-) depth of each element and must be constant over the (x-) length of the element.

4.4 PRESTRESSING TENDON FORMULATION

4.4.1 Tendon Discretization

The prestressing tendon idealization is based on representing the actual curved geometry of a post-tensioned tendon by a system of piecewise linear prestressing segments intersecting at tendon points (**Figure 4.4-1**). Each intersection point is referred to as a *tendon node*. The *tendon nodes* generally do not coincide with the *frame nodes*, which are located at the centroids of elements. Each *tendon node* is associated with a *frame node*. The displacements of the tendon nodes are rigidly locked to the displacements of their associated nodes (**Figure 4.4-2**).

Each tendon segment is assumed to be under uniform axial stress over its length, and its stiffness and load contributions are computed as an eccentric truss element (**Figure 4.4-3**). Initial prestressing forces are computed based on input jacking forces at each tendon end, including the effects of instantaneous friction and anchorage slip losses. The material model for the tendon includes relaxation strains but is limited to linear elasticity for its instantaneous stresses mechanical strain relationship.

During the solution time step when the tendon is initially stressed, the tendon is considered as unbonded and only its load terms are included in the global equilibrium equations. During all subsequent solution steps until the tendon is removed, the tendon is considered as perfectly bonded and both load and stiffness terms are included in the global equilibrium equations. Displacement compatibility between the prestressing steel segments and the frame elements is enforced at the frame nodes to which the tendon nodes are locked. It is handled at the structure level through the direct stiffness assembly procedure.

4.4.2 Tendon Geometry Definition

Prestressing tendon geometry is completely defined by the global X- and Y-coordinates of the tendon points. For tendon geometry nodes, the length of the frame is divided into portions which usually correspond to the actual physical spans of the frame between supports but may be arbitrarily chosen for input convenience.

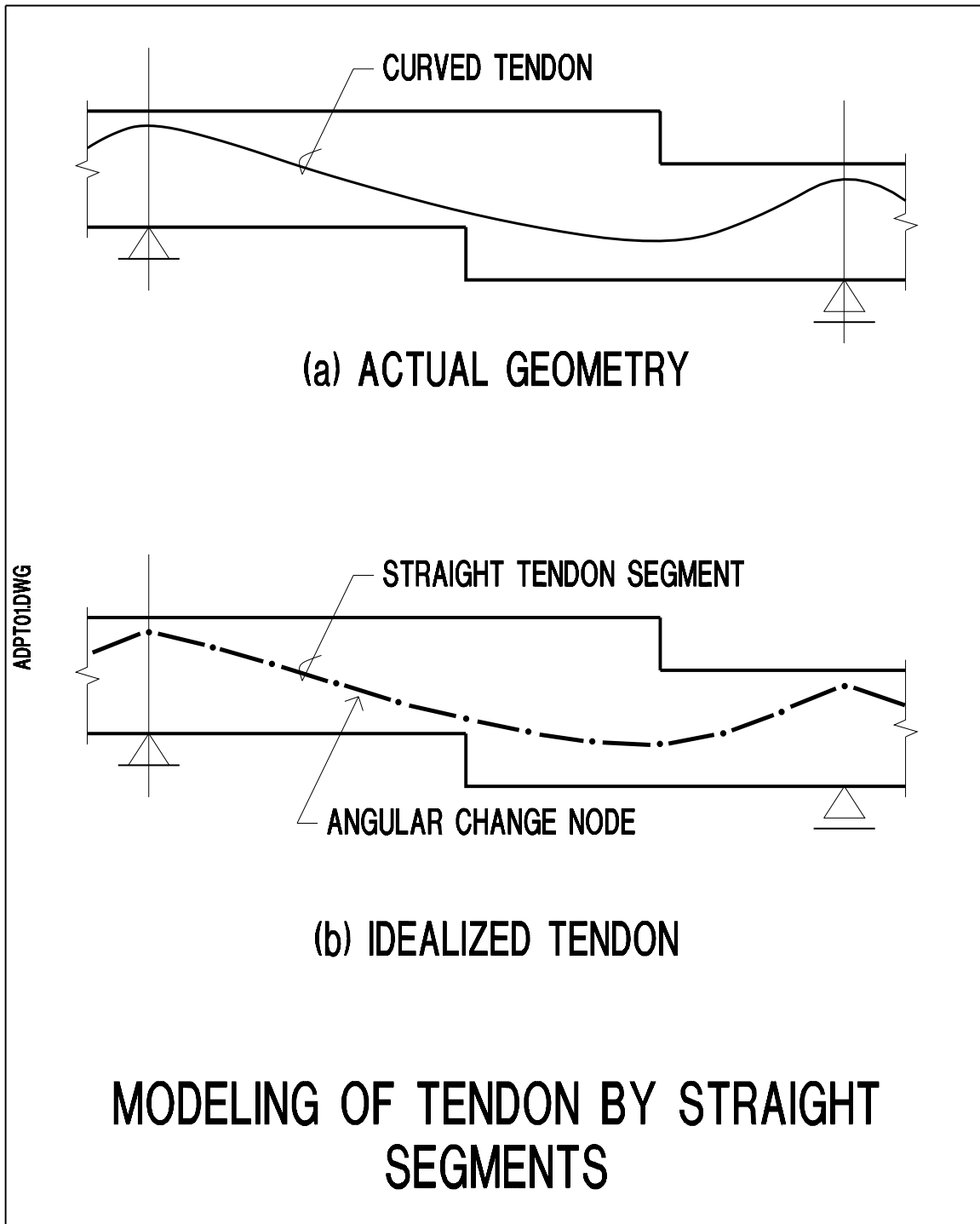


FIGURE 4.4-1

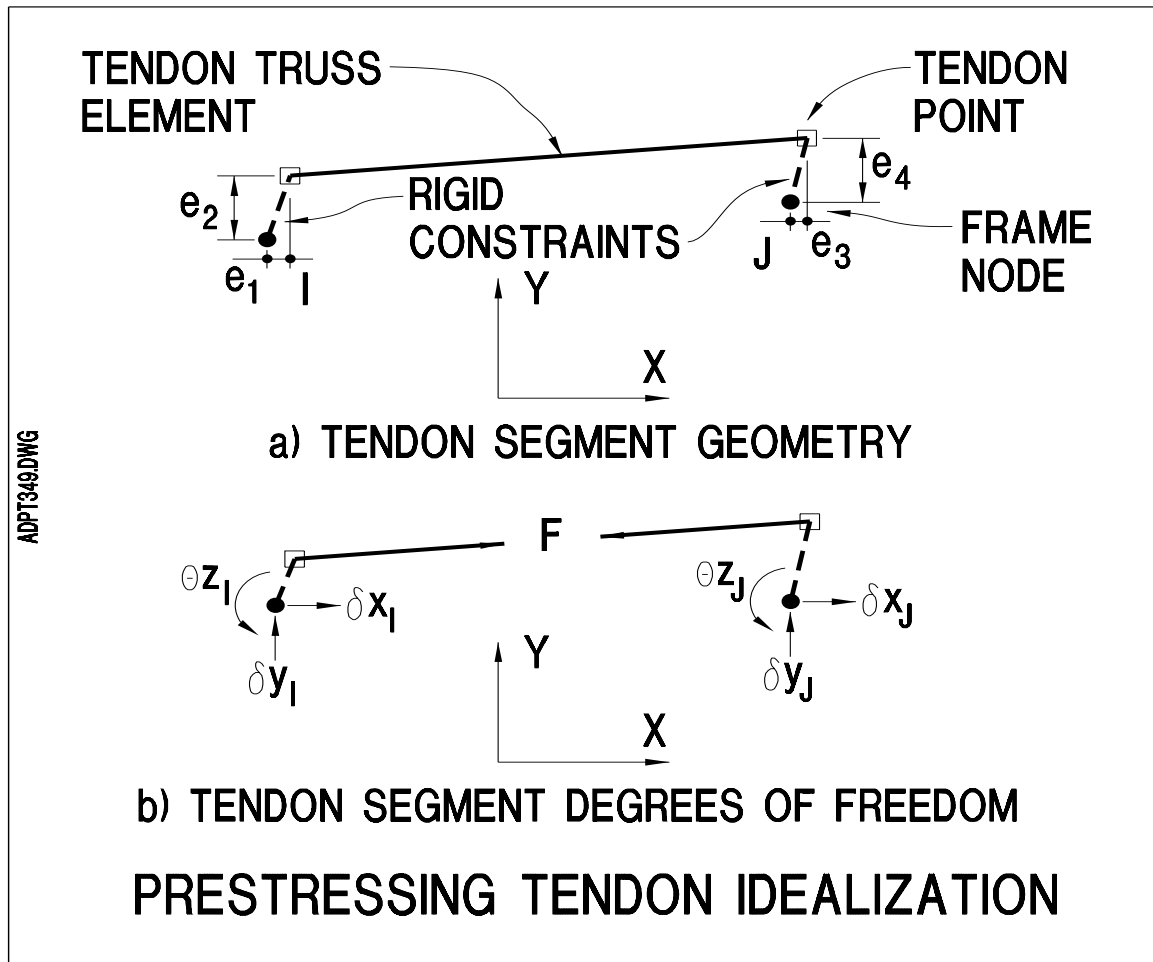


FIGURE 4.4-2

The tendon point coordinates may be specified in any convenient local r- s- coordinate system (**Figure 4.4-2**), which may be defined independently for each span or portion of the tendon. The r- s- coordinates are transformed to the global X- Y- coordinate system using the standard transformation relationships.

For each span of the tendon, the r- and s- coordinates must be either input for each tendon node, or generated using one of the parametric generation schemes provided in the program.

The geometry of each tendon segment (**Figure 4.4-3(a)**) is defined by the two tendon nodes I and J and the eccentricities of these tendon nodes from their associated frame nodes NI and NJ. The tendon node eccentricities in global coordinates are given by:

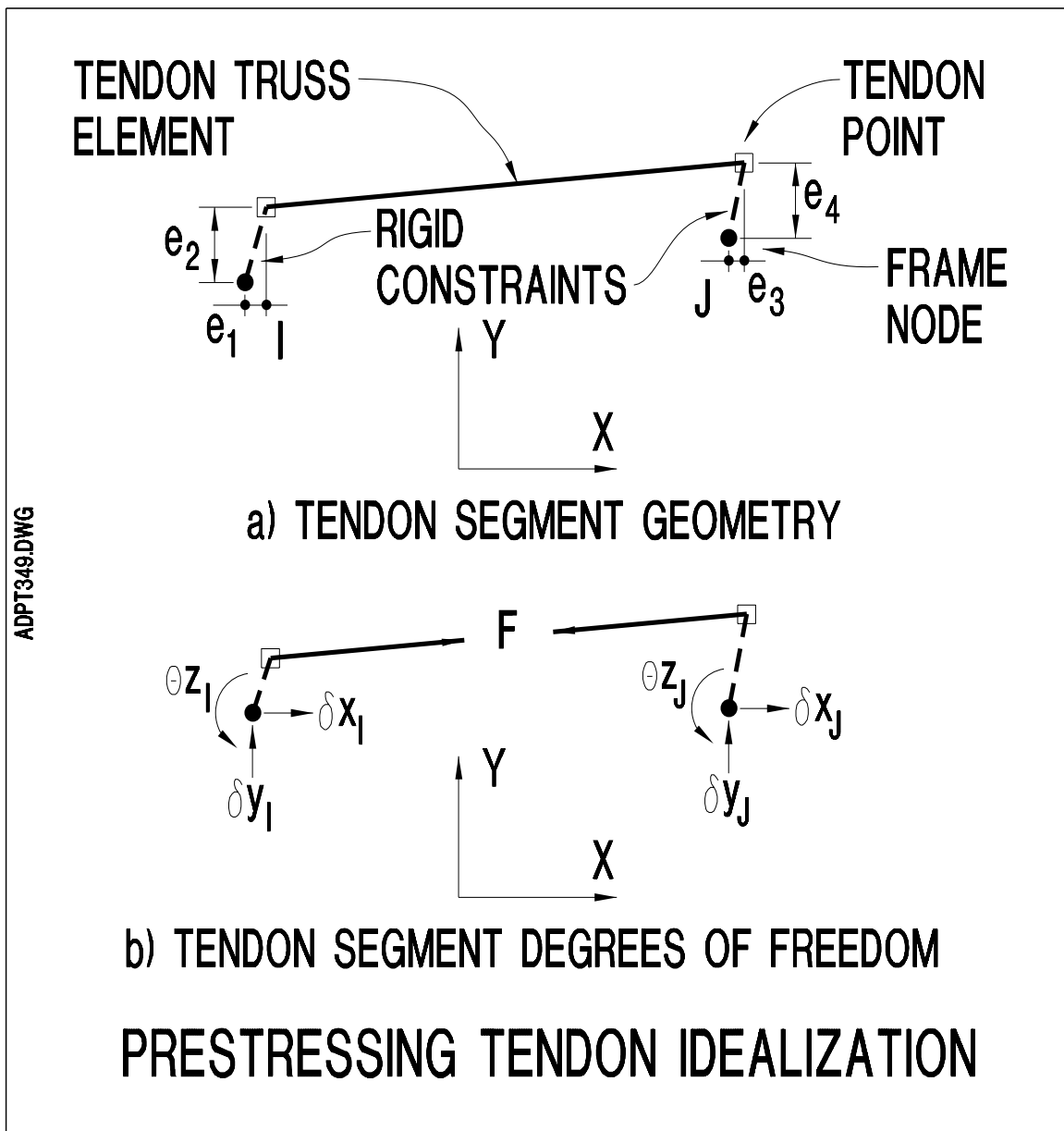


FIGURE 4.4-3

$$\begin{aligned} \mathbf{e}_1 &= \mathbf{X}_{\text{TI}} - \mathbf{X}_{\text{NI}} \\ \mathbf{e}_2 &= \mathbf{Y}_{\text{TI}} - \mathbf{Y}_{\text{NI}} \\ \mathbf{e}_3 &= \mathbf{X}_{\text{TJ}} - \mathbf{X}_{\text{NJ}} \\ \mathbf{e}_4 &= \mathbf{Y}_{\text{TJ}} - \mathbf{Y}_{\text{NJ}} \end{aligned} \quad (4.4.1)$$

Where the suffixes, TI, and, NI, refer to global tendon and frame node coordinates.

Each tendon segment has one local degree of freedom (its internal strain) which is transformed to six global degrees of freedom (the global displacements at the nodes) (**Figure 4.4-3(b)**). The segment element

matrices in local and global coordinates, and the transformation, are discussed in the following sections.

Direct tendon node input is the conceptually simplest approach for specification of tendon geometry. The coordinates of each tendon point must be calculated and input, and for complex tendon geometries this approach can involve a tremendous amount of hand calculation. The parametric generation scheme can perform this task for most typical tendon profiles.

When parametric tendon geometry input is used, the profile of the tendon in a span is specified as a combination of up to four parabolic and/or straight line segments. The tendon node coordinates in the span are then automatically generated based on a parametric description of these four segments. Several combinations of automatically generated tendon geometries are given in **Figure 2.4-1**. It is reiterated that the tendon spans need not coincide with the frame spans. As a result, a frame span can be considered as several tendon spans, each of which made up of four parametrically generated portions.

Given the length, L , of one span of the tendon, the tendon profile is uniquely defined by the following parameters, all normalized with respect to tendon span length, L , (**Figure 4.4-4**):

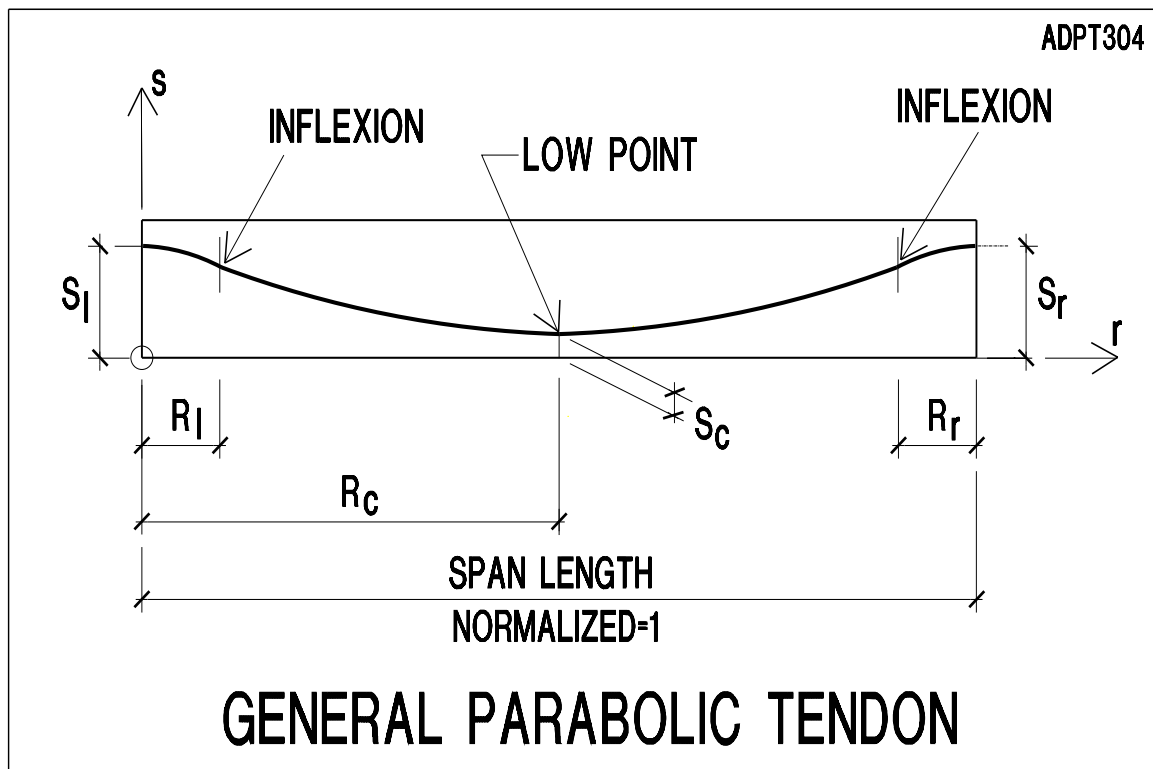


FIGURE 4.4-4

- $0, s_l$ = Coordinates of left tendon end
- r_c, s_c = Coordinates of point of zero slope (low point)
- l, s_r = Coordinates of right tendon end
- r_l = left end to inflexion point
- r_r = right end to right inflexion point

These parameters provide enough information to generate a profile made up of four parabolic segments with zero slope at locations A, B, and C and tangent intersections at r_l , r_c , and r_r . The zero slope at ends, l_v and r_v , provides tangent intersections with neighboring spans.

Many other tendon profiles can be generated. By proper specification of the parameters, most commonly used profiles can be generated. In special cases, it is possible to use more than one portion to represent one physical span of the bridge. This technique allows generation of very complex tendon geometries.

In some cases where non typical tendon geometries are used, it may be necessary to resort to direct tendon point input. With careful selection of tendon portions, and with use of parametric generation in portions where it is possible, the use of direct tendon point input can be minimized.

4.4.3 Determination of Initial Tendon Forces

The initial forces in the tendon segments at the time of the tendon's installation are computed by the program based on input jacking forces at the tendon ends and short term losses over the length of the tendon due to friction and anchorage slip. These losses result in a variation in initial force over the length of the tendon, which is finally idealized by a different initial force in each tendon segment.

A. Friction losses

Due to friction between the tendon and its sheathing during jacking, the tendon force at a distance x from the jacking end is less than the force at the jacking end (**Figure 4.4-5**). This friction loss is considered in two parts: the length or wobble effect and the curvature effect. The length effect is the amount of friction loss that would occur in a straight tendon due to duct imperfections and construction practice, and is dependent on the friction coefficient, tendon length, workmanship and alignment of the duct. The curvature effect results from the intended curvature of the tendon, and is dependent on the friction coefficient and the total angle change between the jacking end and the point under consideration.

The initial tendon force F_x at a distance x from the jacking end can be described by the following commonly used expression:

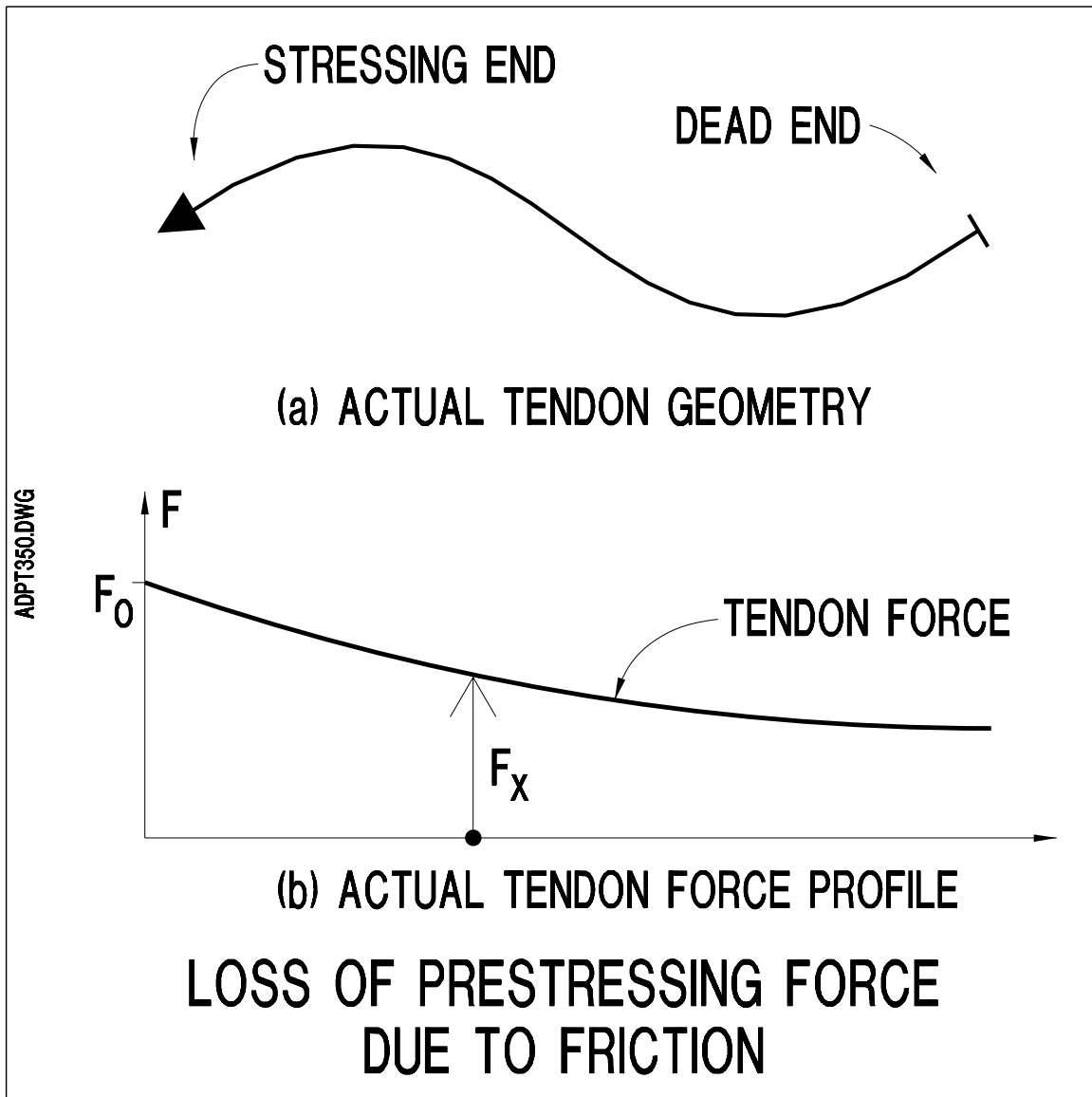


FIGURE 4.4-5

$$F_x = F_0 e^{-(\mu\alpha + Kx)} \quad (4.4.2)$$

where,

- F_0 = Tendon force at jacking end;
- μ = curvature friction coefficient
- α = cumulative angle (in radians) by which the tangent to the tendon profile has changed between the jack and location x
- K = wobble friction coefficient
- x = distance along the tendon

In the piecewise linear tendon discretization used, the angle change α is lumped as discrete θ deviations at the interior tendon nodes and the length x is defined by the tendon segment lengths.

B. Anchor slip (seating) losses

Anchor slip losses take place when the jacking force is transferred from the jack to the permanent tendon anchorage assembly. The anchorage assembly typically displaces up to 3/8 inch during this operation, resulting in a change in stress in a length of the tendon near the anchorage (**Figure 4.4-6**).

The change in stress and the length over which it is effective may be derived from fundamental principles: the friction force reverses direction when the slippage takes place, and the anchor set (slip) displacement, a , must equal the integral of the change in strain over the slip distance XL :

$$a = \int_0^{XL} \delta \epsilon \, dx \quad (4.4.3)$$

This results in the *mirror image* type tendon force change over the anchor slip distance (**Figure 4.4-6**).

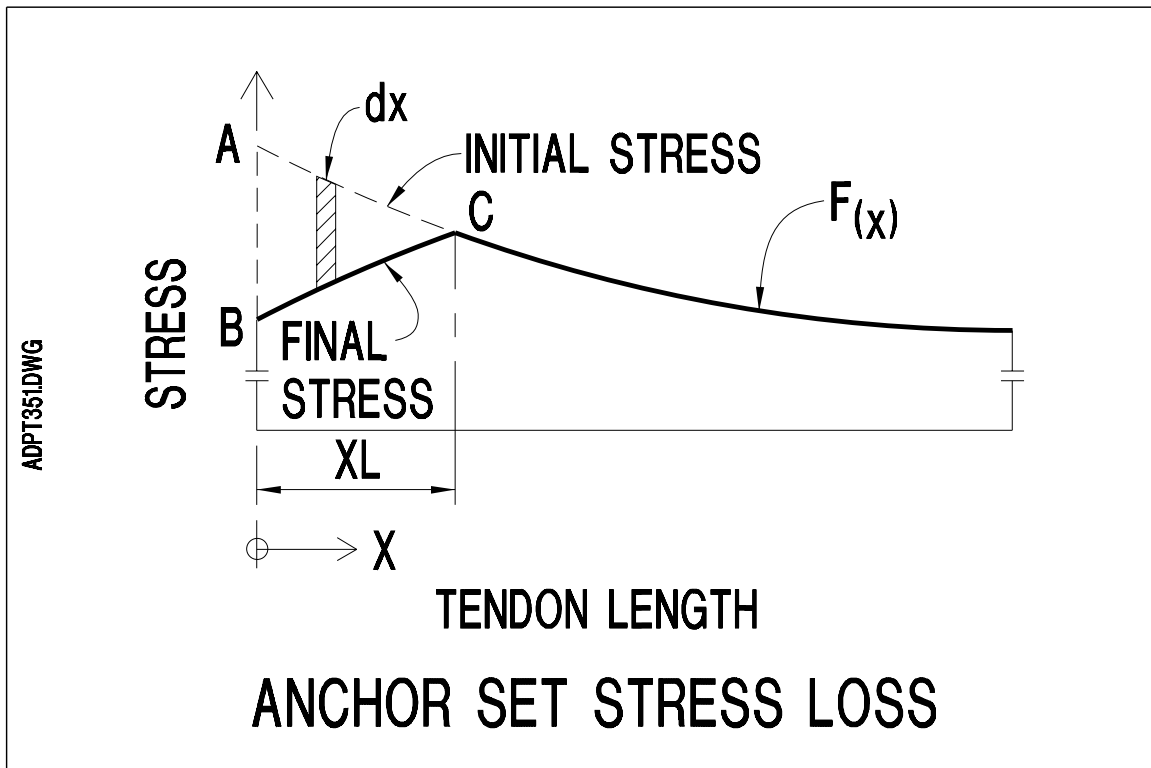


FIGURE 4.4-6

A graphical interpretation of the fundamental principles is taken in order to arrive at the computational approach for determining anchor slip losses. The area of the region between the original force and the force after seating loss (**Figure 4.4-6**) of the tendon force diagram must equal the anchor slip displacement multiplied by, $A \cdot E$, of the tendon:

$$\text{Area of triangle} = a \cdot A \cdot E \quad (4.4.4)$$

Where, A is the area of the tendon and E is its modulus of elasticity.

The tendon segments are scanned starting at the jacking end in order to locate point at distance, XL , (**Figure 4.4-6(a)**) which satisfies this equality. After this point is located, all tendon forces between the jacking end and this point are modified to reflect the *mirrored* force profile.

C. Influence of stressing procedure

In an actual structure, one of several different jacking procedures may be used for each tendon jacking from either one of the two ends or jacking from both ends. The procedure for computing initial forces for one-end jacking is similar for jacking from either end of the tendon; the only difference is the order in which the tendon segments are considered.

For two-end jacking the controlling anchorage must be determined for each tendon point (**Figure 4.4-7**). The procedure used is as follows: First, the tendon point forces are determined under jacking at tendon end, 'A'. Next, the same procedure is initiated for jacking from tendon end, 'B', but as the force at each tendon point is computed it is compared with the force already existing due to jacking from tendon end, 'A'. When it is found that the force already existing in the tendon (from end 'A' jacking) is greater than the force computed for end 'B' jacking, the end, 'B' jacking procedure is terminated. This method closely approximates the physical situation in the actual tendon.

D. Discretized segment forces

After the initial tendon force at each tendon node is computed, the initial force in each tendon segment is found by averaging the forces at its two defining tendon nodes. The average tendon force scheme is applicable if the tendon has a smooth profile. For tendons which are harped and have a rapid change of curvature, finer frame subdivisions must be used, in order to minimize the approximation.

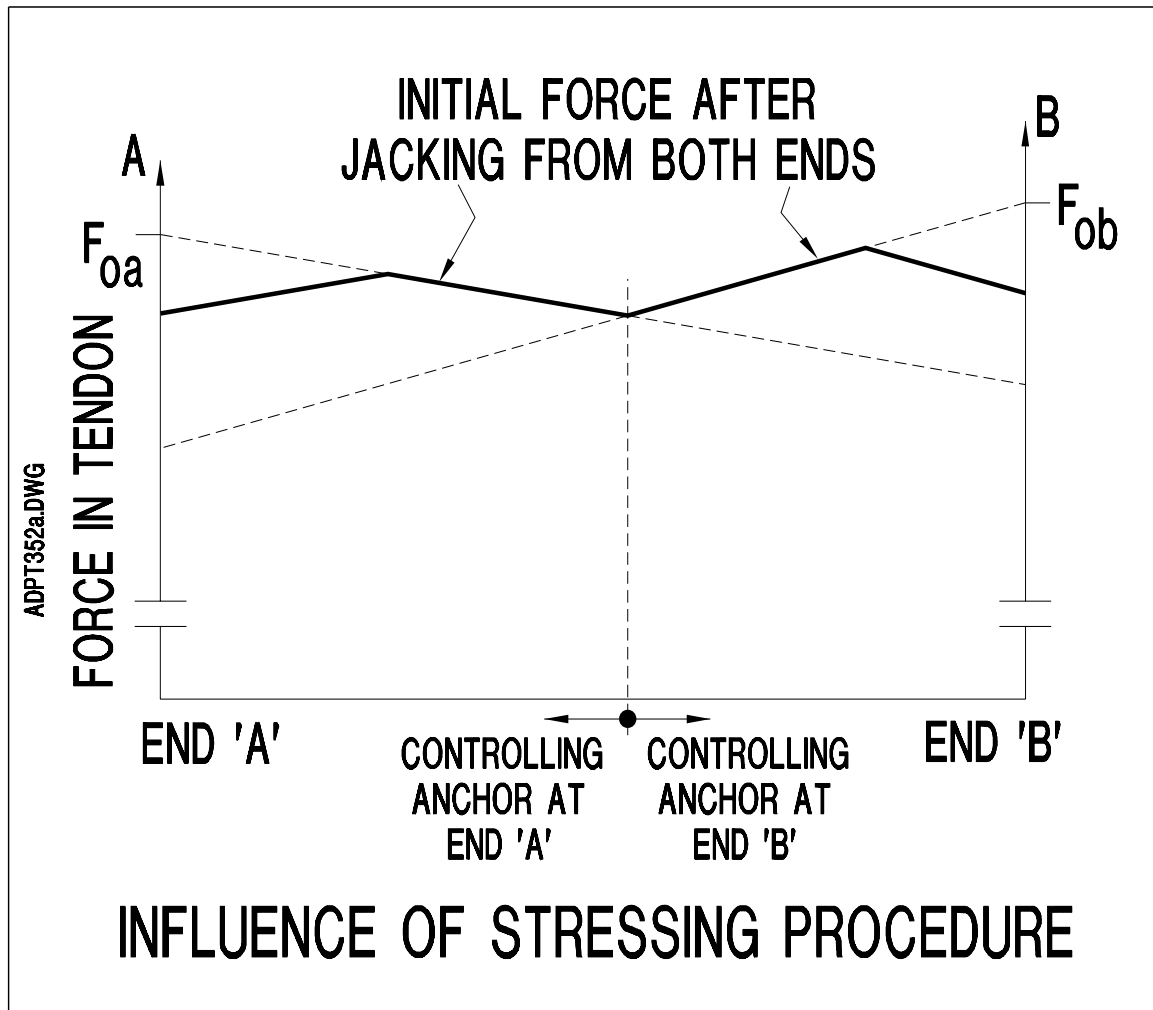


FIGURE 4.4-7

Consider a tendon with a smooth arbitrary tendon profile (**Figure 4.4-8(a)**). The *actual* tendon force profile is a smooth curve (**Figure 4.4-8(b)**). The computed tendon force profile (**Figure 4.4-8(b)**) has curvature friction losses lumped at the tendon nodes and wobble friction losses distributed over the segments. The *best estimate* of the actual tendon force profile from this computed force profile is found by connecting the plotted average force at the tendon node with straight lines (**Figure 4.4-8(c)**). The constant idealized force in each tendon segment can then be found by averaging the segment end force values from this smoothed profile (**Figure 4.4-8(c)**).

This results in a best estimate of the tendon segment forces for a smoothly curved tendon profile. If there are actual concentrated angle changes in the prototype tendon geometry (as found in a harped tendon), then an error may be introduced since no abrupt changes in tendon force are considered. This error is considered inconsequential since it disappears as the tendon discretization is refined, and a minimum radius

of curvature of the tendon is always maintained in the post-tensioned systems for which the program is intended, even when the tendon is *harped*.

E. Equivalent prestressing loads

Once forces in all the tendon segments are known, the equivalent forces acting on the structure at each tendon node may be found. Consider tendon node, i , where tendon segments, $i-1$, and, i , are connected (**Figure 4.4-9**). The vectors from point, i , to, $i-1$, and from, i , to, $i+1$, are v_{i-1} and v_i respectively. These can be expressed as:

$$\begin{aligned} v_{i-1} &= \{x_{i-1} - x_i, y_{i-1} - y_i\}^T; \\ v_i &= \{x_{i+1} - x_i, x_{i+1} - y_i\}^T \end{aligned} \quad (4.4.5)$$

With direction cosines

$$C_{i-1} = v_{i-1}/|v_{i-1}|; \quad C_i = v_i/|v_i| \quad (4.4.6)$$

The components of the tendon nodal forces in the global coordinate system are given by F'_{i-1} and F'_i

$$\begin{aligned} F'_{i-1} &= F_{i-1} * C_{i-1} \\ F'_i &= F_i * C_i \end{aligned} \quad (4.4.7)$$

The resulting equivalent load vector P_i at tendon node i , consisting of F_{txi} and F_{tyi} can be expressed in terms of the two tendon force components F'_{i-1} and F'_i using vector addition.

4.4.4 Material Constitutive Relationships

The prestressing tendon steel is considered as a linear elastic material subject to time dependent strains due to relaxation. Relaxation is a phenomenon similar to creep and is defined as the decrease in stress over time under constant strain. The following equation, proposed by Magura et al (1964) is used for the calculation of tendon stress relaxation.

$$\frac{f_s}{f_{si}} = 1.0 - \frac{\log(t)}{c} \frac{f_{si}}{f_{py}} - 0.55 \quad (4.4.8)$$

where,

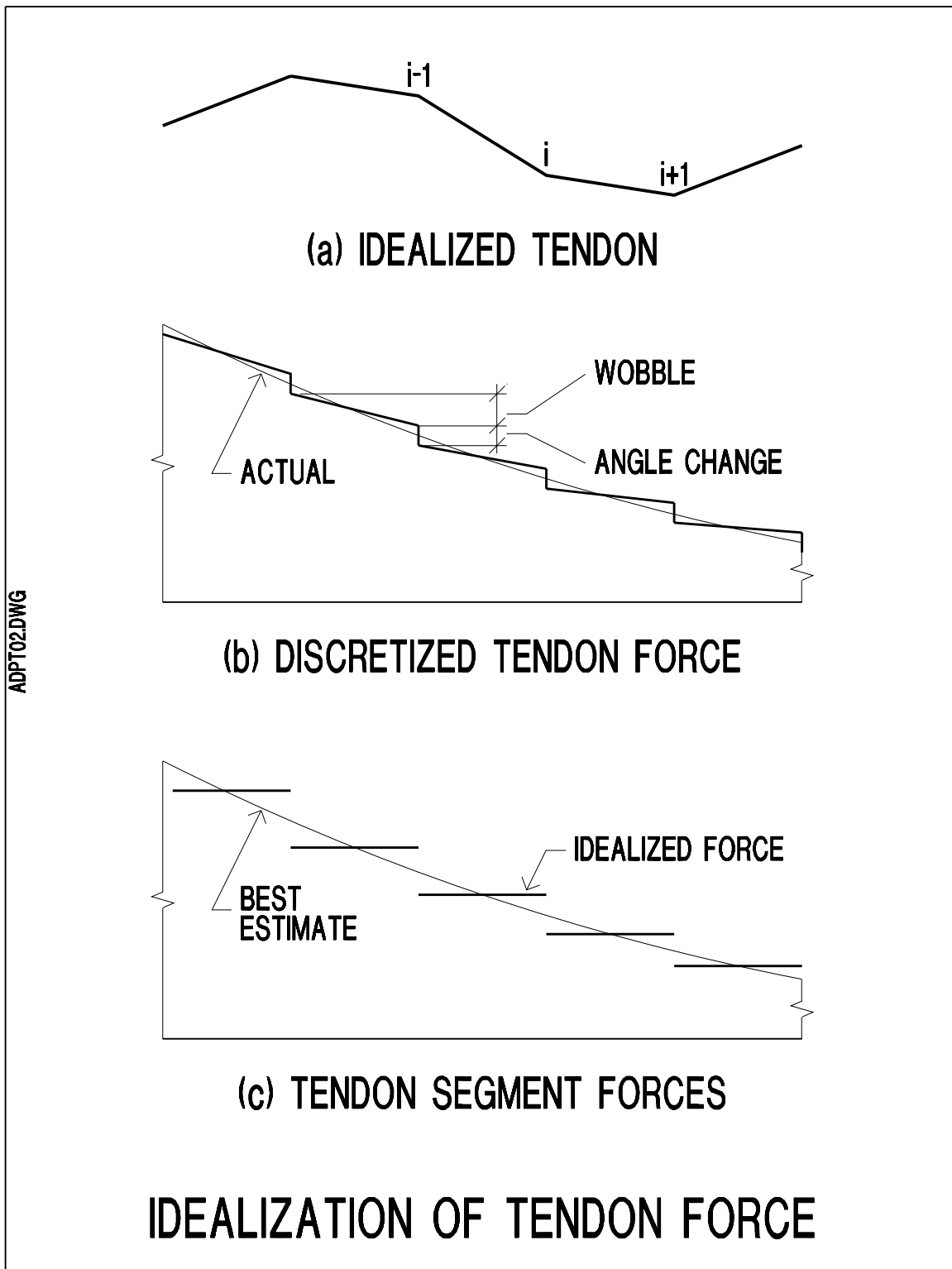


FIGURE 4.4-8

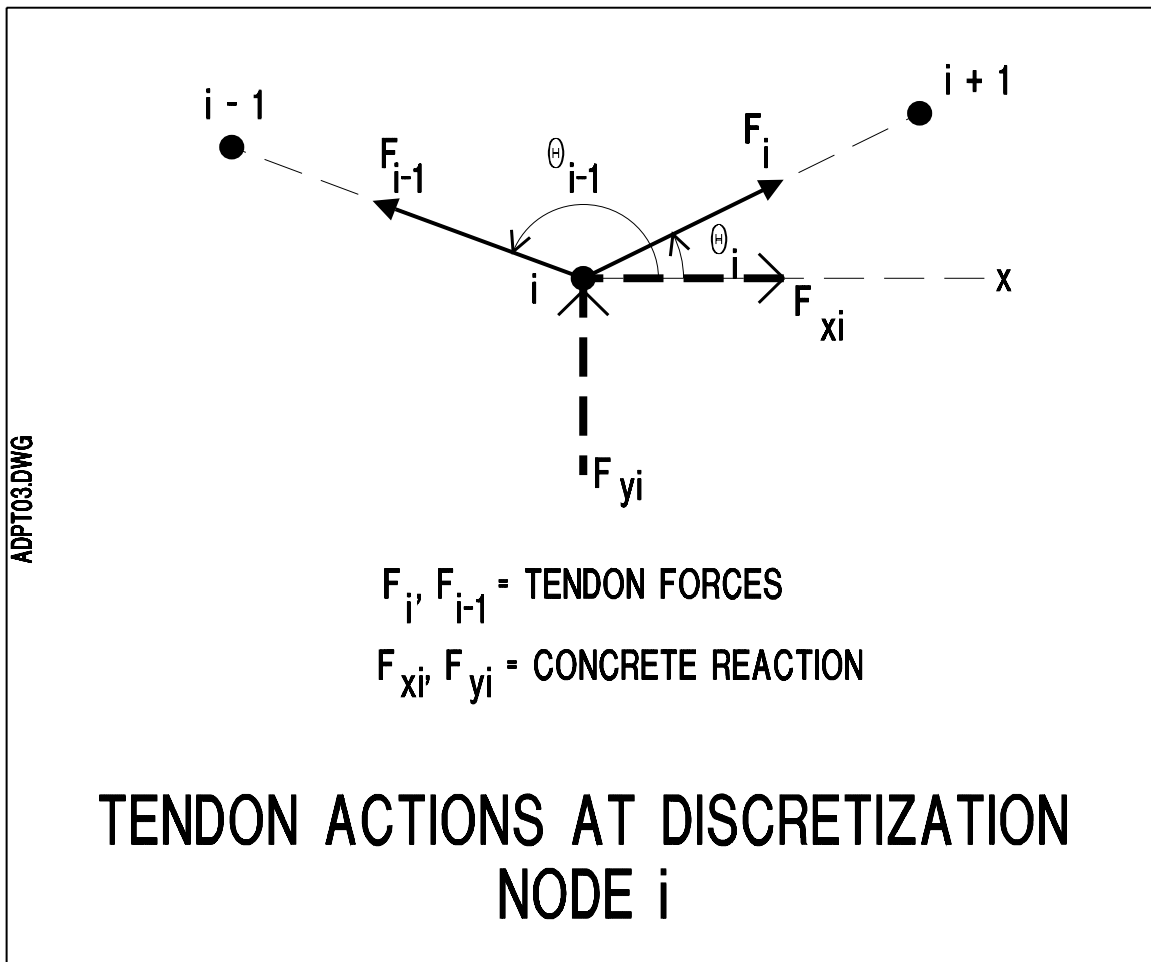


FIGURE 4.4-9

f_s	= steel stress at time t
f_{si}	= initial steel stress
f_{py}	= 0.001 offset yield stress
c	= constant
	10.0 for stress relieved strand
	45.0 for low relaxation strand
t	= time in hours after stressing

Figure 4.4-10 shows a typical variation of prestressing loss due to stress relaxation with time.

This relationship was developed assuming that the strain remains constant and that the only stress applied is the initial prestress. Since creep, shrinkage and external loads cause additional changes in force over time, this Equation 4.4.8 can not be applied directly. To incorporate the additional force variations over time, Hernandez and Gamble (1975) suggested the following procedure which is used in the program, based on

the assumption that all non-relaxation changes in tendon force occur at the ends of the time steps (**Figure 4.4-11**)

The initial tendon force f_{sio} , applied at time t_o , relaxes by the amount δf_{r1} to f_{s1} during the time interval δt_1 :

$$\delta f_{r1} = f_{sio} - f_{s1} \quad (4.4.9a)$$

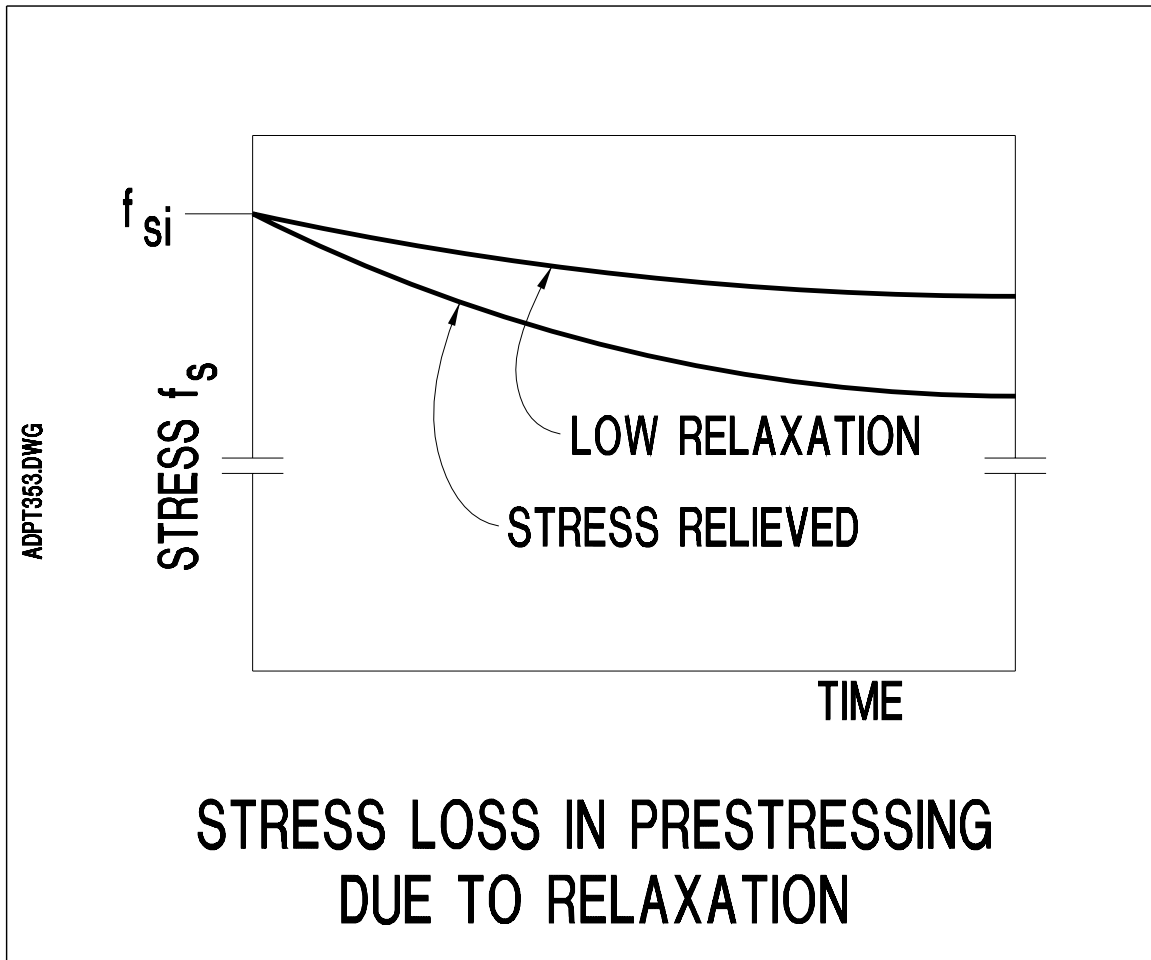


FIGURE 4.4-10

At time t_1 , externally caused strain changes cause the tendon force to change to f_{s1} . To compute the stress relaxation δf_{r2} during the time interval δt_2 , Equation 4.4.9a is used to calculate a fictitious initial tendon force f_{si1} which would have relaxed by δf_{r1} to f_{s1} during δt_1 . The stress f_{s2} may then be found, assuming f_{si1} as the initial tendon force and applying Equation 4.4.11:

$$\delta f_{r2} = f_{si1} - f_{s2} \quad (4.4.9b)$$

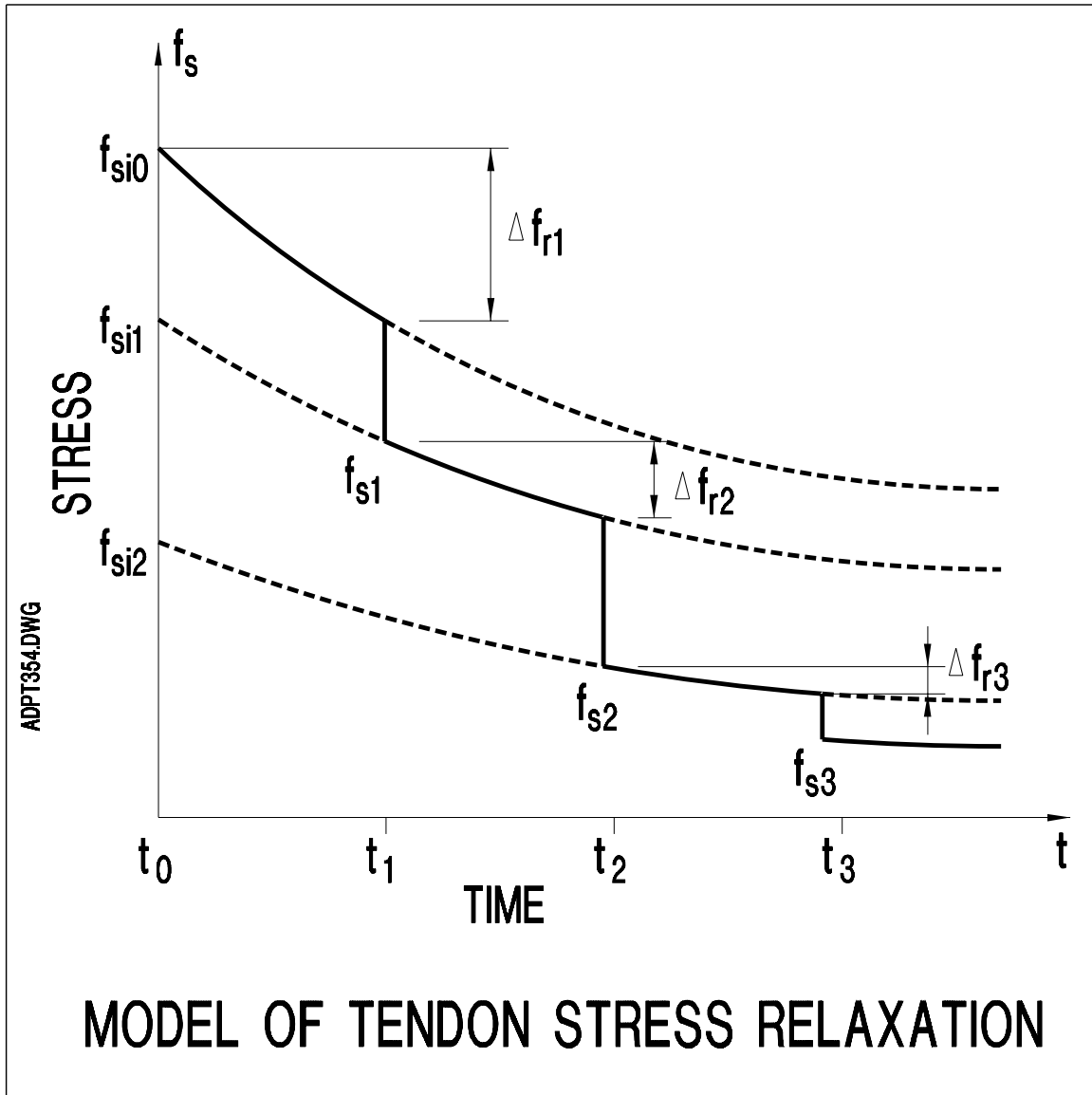


FIGURE 4.4-11

This procedure is applied to each tendon segment during each time step to arrive at the total stress relaxation at time t_n .

4.4.5 Stiffness and Load Computation

During each time step, the following tendon segment element computations are required:

- A. The segment's contribution to the global elastic stiffness matrix $[K]$,
- B. the segment's internal stress due to nodal displacements, and
- C. the segment's equivalent nodal loads due to initial stresses.

During element output, the equivalent segment end stress resultants must be computed for direct comparison with the frame element output. These computations are done using matrix transformations. All the segment element matrices are computed once, prior to the time dependent solution, and are reused in each time step.

The formulation of all the matrices follows elementary linear elastic theory for truss-type elements. This guarantees static equilibrium of computed internal forces with the external loads. **Figure 4.4-3** illustrates dimensions and properties of the tendon segment elements.

A. Segment stiffness matrix

The stiffness matrix $[K]$ for each tendon segment element is evaluated by transformation of the 1×1 segment stiffness matrix $[k]$ in local coordinates by the 1×6 displacement transformation matrix $[A]$ as follows:

$$[K] = [A]^T [k] [A] \quad (4.4.10)$$

where,

$$\begin{aligned} [K] &= 6 \times 6 \text{ global stiffness matrix relating segment global forces to global displacements (**Figure 4.4-3(b)**)} \\ [k] &= 1 \times 1 \text{ local stiffness matrix relating segment local force } F \text{ to local displacements (**Figure 4.4-3(b)**)} \\ &= \langle EA/L \rangle \\ [A] &= 1 \times 6 \text{ displacement transformation matrix relating segment local displacements to global displacements (**Figure 4.4-3(b)**)} \\ &= \{-X/L, -Y/L, e_2X/L - e_1Y/L, X/L, Y/L, e_3Y/L - e_4X/L\} \end{aligned}$$

The segment end eccentricities, e , are defined in Equation 4.4.1.

B. Segment internal force due to nodal displacements

The segment internal force increment is evaluated by first transforming the global displacements (**Figure 4.4-3**) to local segment element displacement, and then multiplying this displacement by the local segment element stiffness matrix $[k]$ defined above. This transformation can be expressed as follows:

$$\{\sigma\}^T = [E] \{U\} \quad (4.4.11)$$

where,

$$\{\sigma\}^T = \text{segment internal force}$$

$\{U\}$ = global displacements at the nodes (**Figure 4.1-1(c)**)
 $[E]$ = displacement to force transformation matrix (1 x 6),
 expresses segment internal force F in terms of global
 displacements
 $= [k] [A]$
 $[k]$ and $[A]$ are defined in the preceding.

C. Equivalent nodal loads due to initial forces

The equivalent nodal forces and moments due to segment internal force are evaluated using the following matrix transformation:

$$S = [A]^T \{\sigma\}^T \quad (4.4.12)$$

where,

S = global equivalent nodal loads
 σ = internal segment force
 $[A]$ = 1 x 6 displacement transformation matrix defined above

The 6 x 1 internal force to global equivalent load transformation matrix is equal to the transpose of the displacement transformation matrix $[A]$ as can be shown by elementary structural theory.

4.5 TRAVELING FORMWORK FORMULATION

4.5.1 Traveler Function and Purpose

Traveler is a frame independent from the primary bridge, or other frame structure being analyzed. It can be moved and attached to the primary frame. It can serve principal purposes. (i) It can be used to transfer the weight of newly cast concrete to the primary frame, as it is done in the prototype construction through the falsework - referred to in this work by the umbrella term of *traveling formwork*. (ii) It can be used to stiffen, or align the structure temporarily during the construction.

The geometry of the traveler is independent from that of the primary frame. It is defined independently and is considered pinned to the primary frame at frame nodes.

The traveler element is provided in order to restrain the displacements of freshly cast frame elements which have zero or near-zero elastic modulus and which would otherwise undergo large incremental displacements which are prevented by the actual traveling formwork. The program also automatically generates the loadings required to model the moving of the traveler.

Each traveler consists of a number of beam elements (**Figure 4.5-1(a)**) which can be moved to any location in the structure during the solution process. The beam element is based on Bernoulli-Euler beam theory and a linear elastic material model. This allows the approximate modeling of many types of formwork or moving auxiliary structures, which may be fixed to the structure during construction.

Whenever a traveler is moved to a new location, its element matrices and loads are recomputed for the nodal geometry of its current location. These matrices and loads are then summed into the equilibrium equations during subsequent solution steps. They are automatically removed from the summation when the traveler is again moved.

4.5.2 Traveler Geometry

Each Traveler (**Figure 2.6-1**) is made up of a number of traveler frame elements. Traveler element geometry is defined the same as frame element geometry (**Figure 4.5-1(b)**). Each traveler element is defined by the two nodes I and J located in the global X- Y- plane and on the centroidal axis of the traveler element. The centroidal axis of the traveler element is assumed to coincide with the centroidal axis of any frame element connected to the same nodes. In most cases, this is a reasonable approximation of the actual traveler. The origin of the local coordinate system is at node I. The local x- axis is defined by the vector joining node I and node J. The local z- axis is parallel to and in the direction of the global Z- axis. The local y- axis is orthogonal to the local x- and z- axes and is directed according to the right hand rule.

The element cross section is defined in the local y- z- plane and must be constant over the (x-) length of the element. The only constraint on the shape of the cross section is that it must be symmetric about the local y- axis. Any arbitrary cross section which meets this requirement can be modeled. Any other type of structure, such as the truss type travelers frequently used, can be modeled approximately using these elements.

4.5.3 Material Constitutive Relationship

The travelers are considered to consist of a linear elastic material. Its constitutive relationship is given by

$$\sigma = E \epsilon \quad (4.5.1)$$

where,

$$\begin{aligned} \sigma &= \text{total stress at time } t_j \\ E &= \text{constant elastic modulus} \end{aligned}$$

ϵ = total strain at time t_j

Temperature strains are not considered in the traveler elements.

4.5.4 Stiffness and Load Computation

During each time step, the following element computations are required:

- A. The element's contribution to the global elastic stiffness matrix [K],
- B. the element's internal forces due to nodal displacements, and
- C. the element's equivalent nodal loads due to initial strains and forces.

These computations are done using matrix transformations.

The total weight of the traveler is assumed constant, regardless of its location in the primary structure. When the traveler is installed at a new location, its equivalent dead loads are computed based on the total weight of the traveler and the tributary contribution of the traveler's geometry to each frame node (**Figure 4.5-1(b)**)

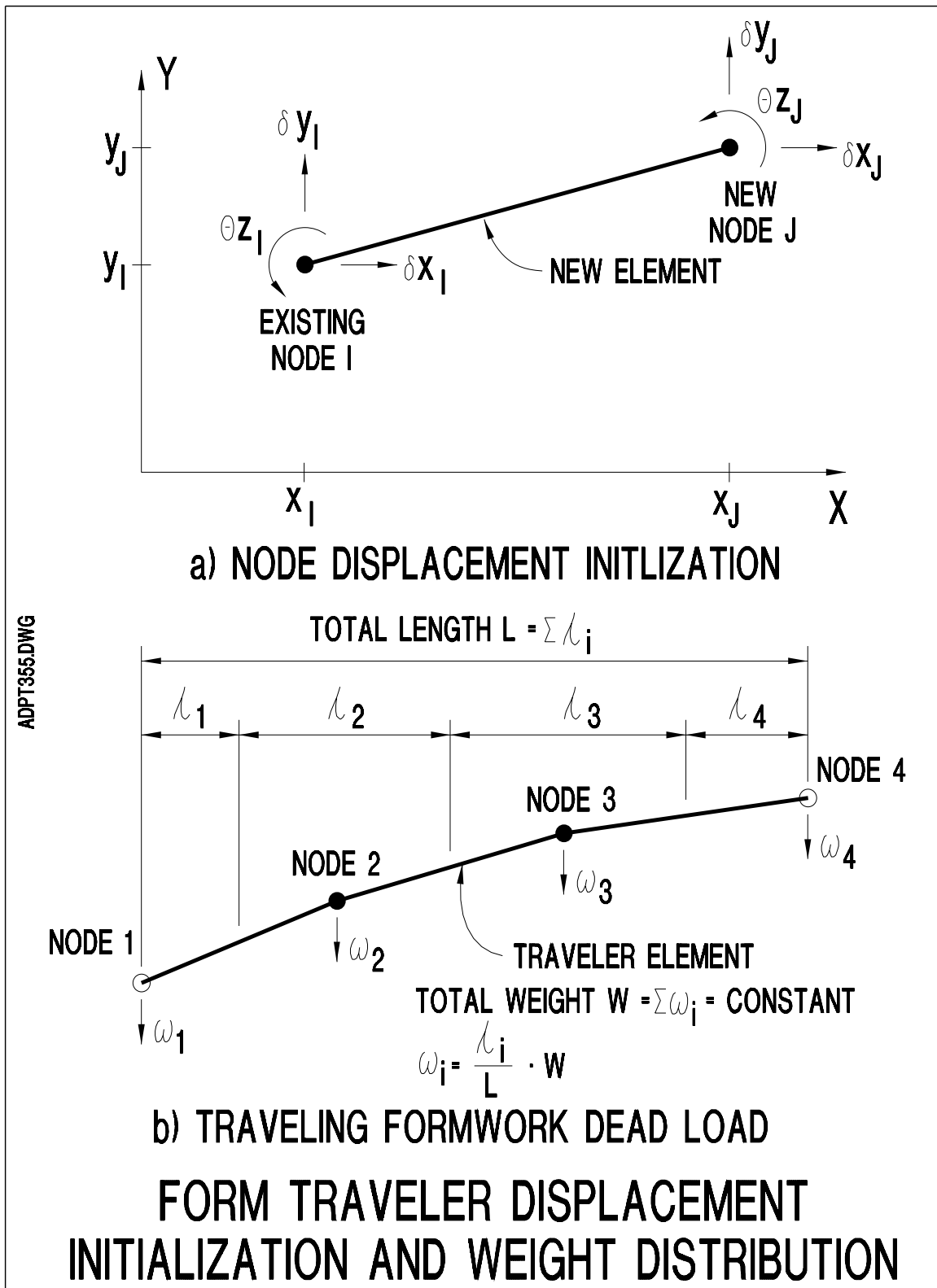


FIGURE 4.5-1

This Page Left Intentionally

BLANK

5. **EXAMPLES - VERIFICATION**

5.1 CANTILEVER CONSTRUCTION BOX GIRDER BRIDGE EXAMPLE 3

5.2 BRIDGE PARTICULARS.....3

5.3 ANALYTICAL MODEL 9

5.4 LOADINGS & CONSTRUCTION SEQUENCE FOR THE TIME DEPENDENT
 ANALYSIS 12

5.5 COMPUTER INPUT 14

5.6 COMPARISON OF RESULTS 24

5.7 AMERICAN, SI AND MKS SYSTEM OF UNITS EXAMPLE 26

5.8 STRUCTURE 26

5.9 RESULTS 27

This Page Left Intentionally

BLANK

This Chapter includes two examples. The first example is a prototype bridge, selected to illustrate details of input generation, modeling, and application of the software to a practical problem. Also, since the first example is adapted from the available literature, it provides a good basis for comparison of ABI solutions to other software. The second example is a simple prestressed structure devised to illustrate the application of the software in different systems of units (American, SI and MKS). It demonstrates the correlation of the solution among them.

Numerous examples illustrating the many facets of the software are given in the *Examples Manual* which is compiled in a different volume.

5.1 CANTILEVER CONSTRUCTION BOX GIRDER BRIDGE EXAMPLE

This example is an adaptation from [Ketchum, Scordelis, 1986]. It is presented with two objectives in mind. First, it demonstrates the capabilities of ADAPT-ABI for analysis of a realistic multi-span prestressed concrete box girder bridge using cantilever construction method, cast-in-place segments, traveling formwork, and conventionally erected girder segments near the abutments. Second, it serves as a basis for comparison of results obtained from ABI with two other software from which analysis solutions for this example are available [Scordelis, 1993].

The structural behavior of the bridge under dead load, construction loads and prestressing is traced through the construction phase and over a 27 year service period thereafter.

The input data for the computer analysis is prepared to match (Ketchum, Scordelis, 1966) as closely as practical. Taking advantage of the symmetry of the problem, only one-half of the bridge is modeled (**Figure 5.3-1** and **5.6-1**). The pier and pier table region are modeled identical to previous solutions (**Figure 5.6-2**). For a regular ABI analysis, a slave element would have been used for connection between the pier top and pier table centroid.

The input data used to generate the ABI solutions is given in Section 5.5. The results of ABI are compared with the other available solutions in Section 5.6.

5.2 BRIDGE PARTICULARS

The bridge chosen for this example and verification (**Figure 5.2-1**) is a straight, three span, single cell haunched box girder bridge. The haunched girder is cantilevered from the piers using cast in place segments, and is later made continuous with a short, conventionally erected, cast in place girder near the abutments and with the adjoining cantilevered girder at midspan. Each cantilever segment is post tensioned to the previous

segments with several *cantilever* tendons, and after the closures at the abutments and at midspan the entire bridge is prestressed with several additional *continuity* tendons extending the full length of the bridge and *local* tendons in zones of high positive moment. This is a common construction sequence and prestressing scheme for bridges of this type. The details of the design, the design criteria and the construction sequence are discussed next.

The span arrangement consists of a 450 foot center span and two 285 foot side spans (**Figure 5.2-1(a)**). This provides for both efficient cantilever construction and satisfactory behavior of the continuous bridge. The cantilevered girders are symmetrical about each pier, and the 60 foot cast in place girder lengths at the abutments help eliminate interference between the traveling formwork and the abutment and also result in better structural behavior of the completed structure under superimposed loads and live load.

The cross section (**Figure 5.2-1(b)**) consists of a single cell box with wide cantilevered slabs. Transverse slab tendons are required in the top slab, but are not included in this analysis. The girder depth varies from a maximum of 24 feet at the piers to a minimum of 9 feet at midspan and at the abutments. The bottom slab thickness varies from a maximum of 3.5 feet at the piers to a minimum of 0.75 feet at midspan. The variations of these dimensions over the length of the bridge are summarized in **Figure 5.3.1(c)** and **5.3.1(d)**. The depth of the girder at any point is found using an expression of the form $d = a(225-x)^{1.7} + d_0$ where, x , is the distance from the supporting pier, d is the girder depth at location, x , and, a , d_0 are constants found by evaluating the expression at, $x=0$ and, $x=225$, where the girder depths are known. Due to the sloping webs and haunched girder, the bottom slab width varies from a maximum of 29 feet at midspan and the abutments to a minimum of 24 feet at the piers. All other cross section dimensions are constant over the full length of the bridge. The details of the dimensions of the bridge may be found by examination of the input listing for the analysis given in Section 5.5.

Four *cantilever* prestressing tendons, located in the top slab, are anchored at the end of each girder segment during the construction phase (**Figure 5.2-2(b)**). This results in a final total of 64 tendons in the top slab over the piers, dropping off uniformly toward center span (**Figure 5.2-2(c)**). Each cantilever tendon consists of 21-1/2 inch diameter strands and is stressed from both ends.

Eight *continuity* prestressing tendons, extending over the full length of the bridge with a draped profile in the webs, are stressed after the bridge is made continuous with mid span closures (**Figure 5.2-2(d)**). The center span continuity tendons are stressed from both ends at coupler anchorages over the piers, and are then coupled to the side span continuity tendons which are stressed from the abutment end only. Each continuity tendon consists of 21, 1/2 inch diameter strands. The continuity tendon between the abutment and the coupler over the first pier is modeled as one tendon (see T17, **Figure 5.6-3**). The

continuation of the continuity tendon from the coupler to the centerline of center span is modeled as a second tendon (see T18, **Figure 5.6-4**).

Twenty-four *local* tendons in the center span and eight *local* tendons in the side spans, located at the bottoms of the webs, are also stressed after the bridge is made continuous (**Figure 5.2-2(e)**). The distribution of these tendons is shown in **Figure 5-2(f)**. Each local tendon consists of 21, 1/2 inch diameter strands and is stressed from both ends.

Design loads on the girder include structural dead load based on 155 pcf concrete, a superimposed dead load of 2.5 kips/foot, and AASHTO HS20-44 lane loading.

The design criteria and method used for proportioning the cross section and prestressing may be summarized as follows [Ketchum, Scordelis, 1986]:

The cross section dimensions and top slab prestressing were proportioned based on an ultimate strength analysis of the statically determinate cantilever girder under total dead load, AASHTO lane live load and prestress. The draped continuity tendons and the local tendons were proportioned based on an allowable stress analysis of the final, continuous system under total dead load, live load and prestress. Then, using this preliminary design as a model, the design of the tendons was finalized based on a *design redistribution* analysis under total dead load, live load and prestress, assuming 100 percent moment redistribution. For checking the adequacy of the design based on this analysis, an allowable stress approach was used, with 0 psi tension and 2250 psi compression allowed under total dead load, live load and final prestress.

The *design redistribution* analysis for the 100 percent redistribution case was performed with the software, using a one step construction and linear elastic analysis sequence in which all concrete segments were assumed to be cast on the average of their actual casting dates, and the total dead load and prestressing were applied at the time of closure. The long term prestressing force losses were estimated at seventeen percent based on this construction schedule and the AASHTO code relationships for estimating prestressing losses. The material properties and tendon stressing data for this analysis were the same as for the detailed segmental analysis described herein, with jacking forces and anchorage slip reduced seventeen percent to allow for long term force losses.

Additional information on analysis procedure and design considerations of cantilever segmental construction is given in [Aalami 1993].

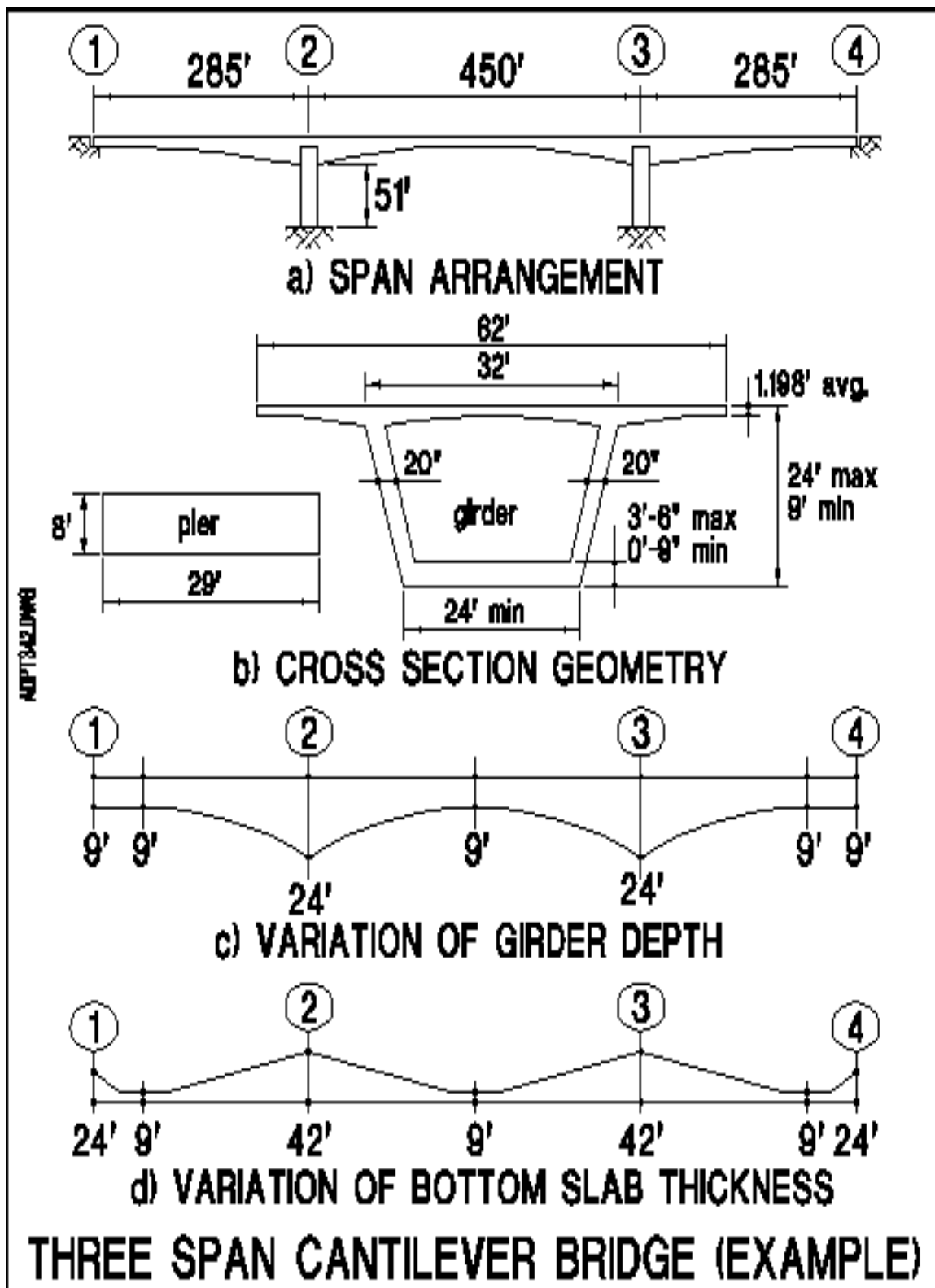


FIGURE 5.2-1

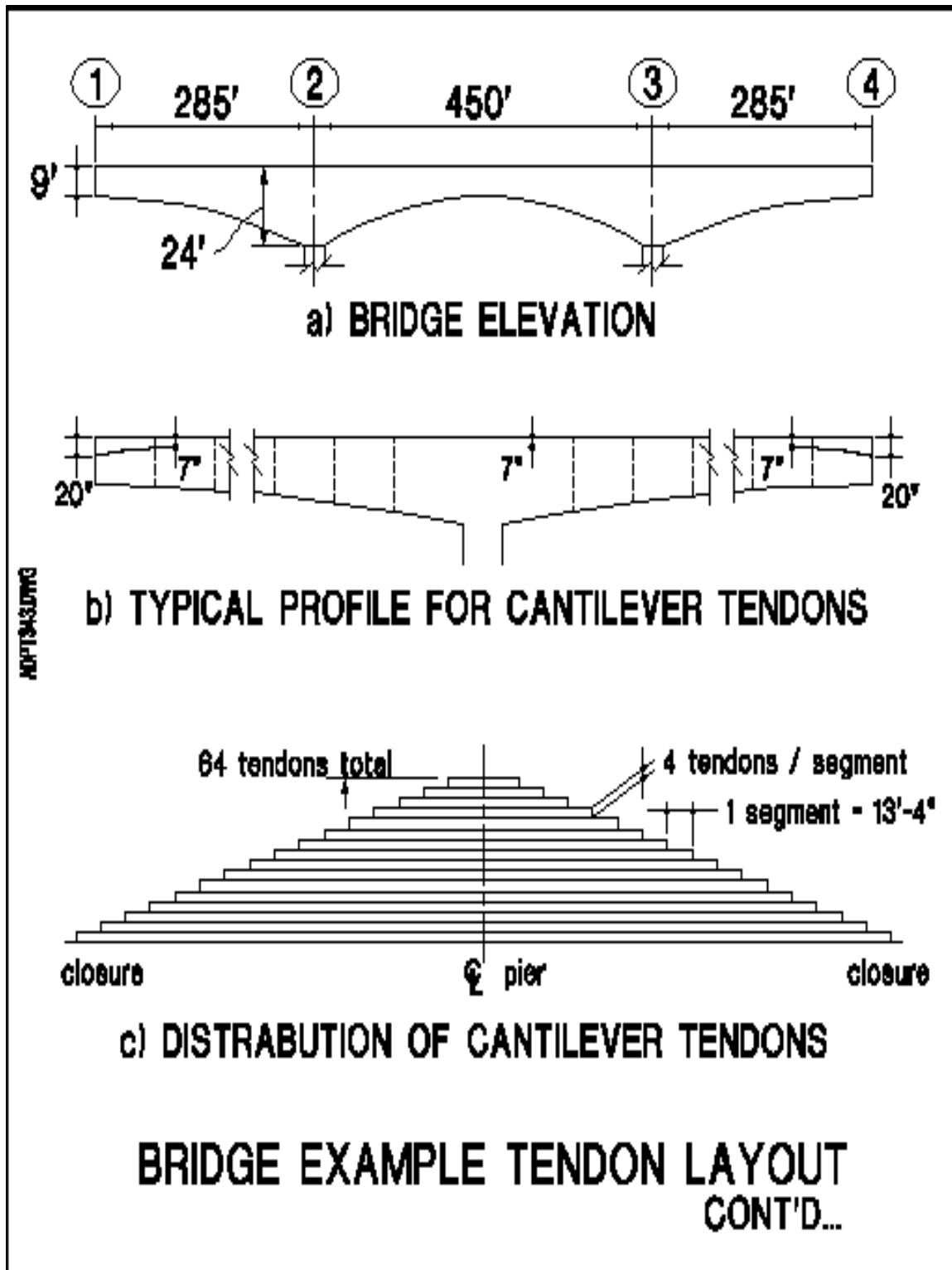
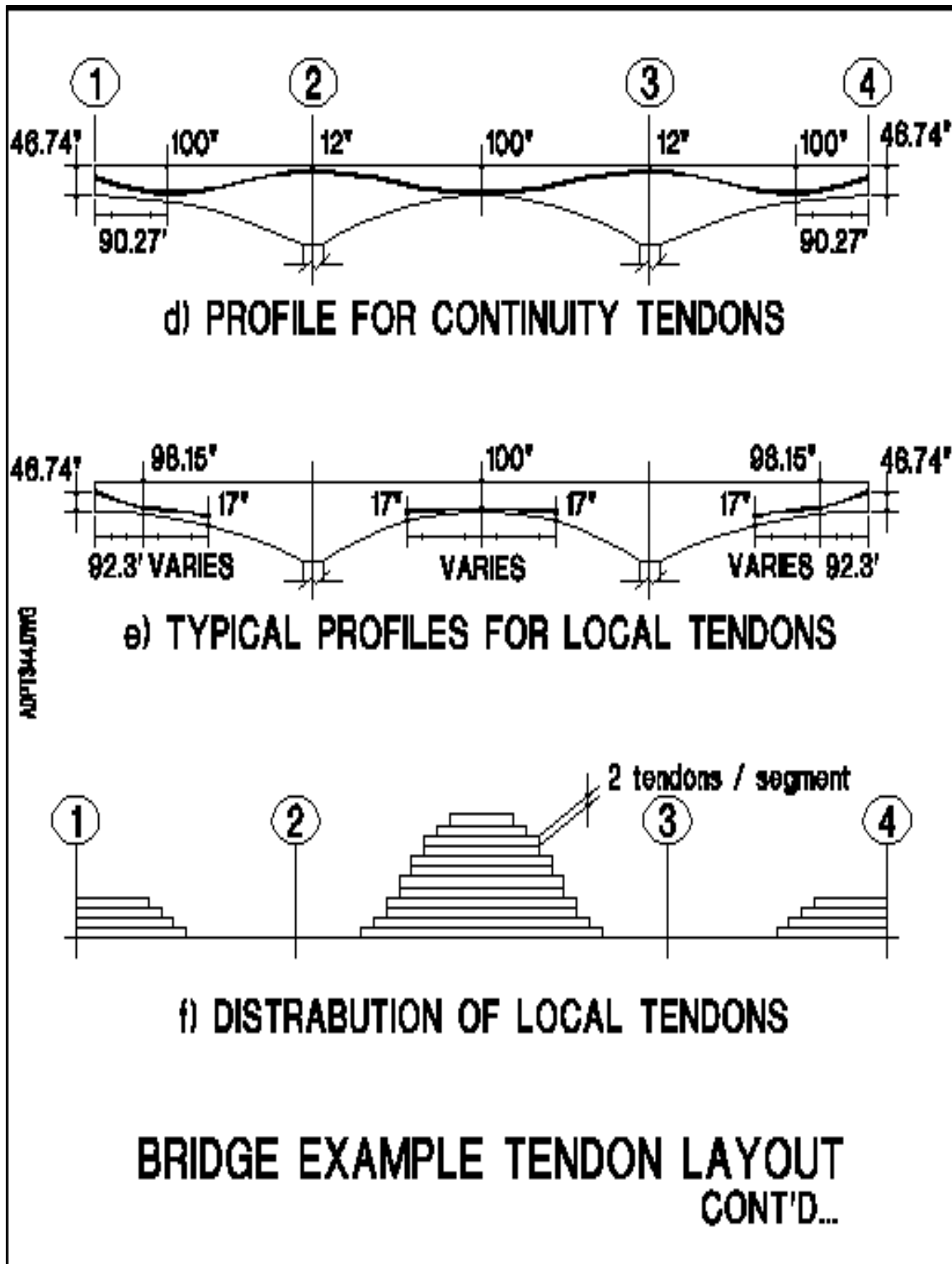


FIGURE 5.2-2 continued ...



continued ...FIGURE 5.2-2

5.3 ANALYTICAL MODEL

The analytical model for the complete time dependent analysis with the software (**Figure 5.3.1**) is of a symmetrical half of the structure, includes the pier and girder, and consists of 43 nodes, 42 frame elements and 30 prestressing tendons. Units used in the analysis are inches and pounds.

The analytical model selected is identical to [Ketchum, Scordelis, 1986] to afford comparison. The model as generated in the computer is shown in **Figure 5.6-1** and **5.6-2**. For the ABI analysis a slave element between the pier top and the pier table centroid is recommended. However, none is used herein to keep the two models similar.

The nodes (**Figure 5.3.1(a)**) are located at segment joints along the centroidal axis of the box girder. Additional nodes are used in the 60 foot cast in place girder length at the abutment and near the pier in order to accurately model the prestressing. The nodes are numbered for ease of node and element generation, without regard for stiffness matrix bandwidth minimization. The SEQUENCE command is then used to specify the internal degree of freedom numbering order for the nodes, in order to minimize the stiffness matrix storage and numerical effort required in the solution of the equilibrium equations.

All nodes other than nodes 1, 41 and 43 are free to displace in all degrees of freedom. Node 1, at the abutment, is restrained against displacement in the global Y direction, representing a sliding bearing support. Node 41, at center span, is restrained against displacement in the global X direction and rotation about the global Z axis, representing the symmetry condition required for modeling half the structure. Node 43, at the bottom of the pier, is restrained against displacement in all three degrees of freedom, representing a foundation on bedrock. Nodes 2 through 5 are temporarily restrained against displacement in the global Y direction, in order to represent the formwork supporting this conventionally erected girder segment at the abutment. These temporary restraints are released after the closure segments are cast and the local tendons are stressed.

In a rigorous ABI analysis, the conventionally erected girder segment at the abutment would be modeled with segments resting over falsework during the construction phase. The falsework will be modeled by spring elements representing the stiffness of the falsework. After the closure between the cantilever section and the abutment girder, and after stressing the continuity tendons, the spring elements representing the falsework would be removed. Spring elements were not used in the analysis offered herein, in order to retain similarity among the three solutions used for comparison.

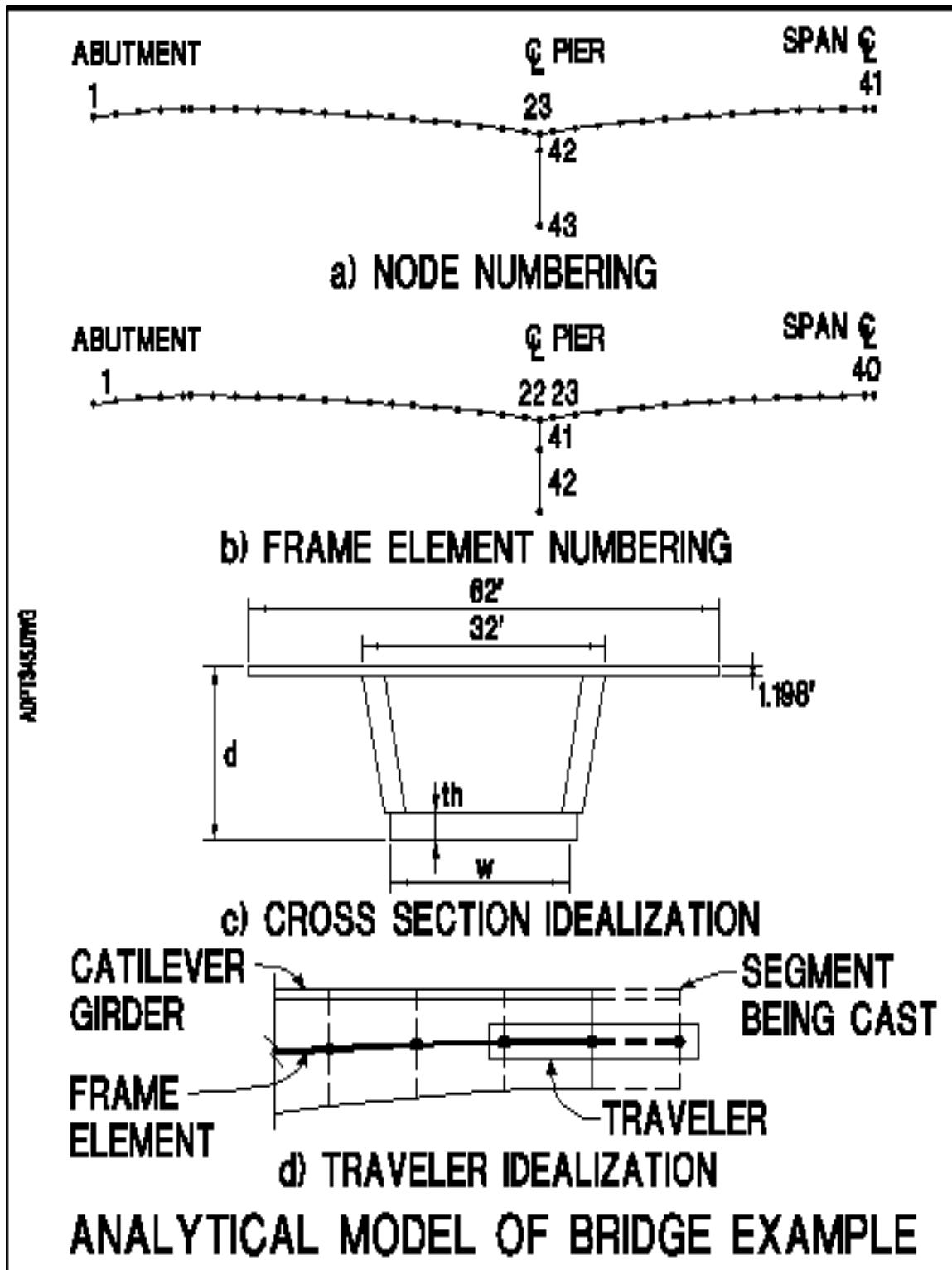


FIGURE 5.3-1

Frame elements (**Figure 5.3.1(b)**) are used to model the girder and pier. The elements modeling the girder are each prismatic with the cross section of a point at the mid length of the element. **Figure 5.6-1** is an illustration of the frame elements used. Each cantilever segment is modeled with one frame element. Additional elements are used to model the closure segments, the 60 foot girder segment at the abutment, and the pier. For section property and dead load generation, the cross section of each girder element is idealized as shown in **Figure 5.3.1(c)**. The pier elements model the gross cross section of the pier. Uniformly distributed mild steel reinforcement is included in all elements, based on an area ratio of 2 percent in the girder and 10 percent in the pier.

Traveling formwork for construction of the concrete segments is included by modeling each actual traveler as two short beam elements. For the construction of each segment, the traveler is moved so that it supports the new concrete segment (**Figure 5.3.1(d)**). Each traveler is assumed to weigh 150,000 pounds.

Tendon elements are used to model the cantilever, continuity and local prestressing tendons described in foregoing. The 30 tendons used to model all the prestressing include a total of 442 tendon segments.

Tendons 1 through 16 each represent four cantilever tendons anchored in each segment. These tendons are straight over most of their length, except for a 13 foot length at each end where they drape to their anchorage locations. Each tendon is jacked from both ends with a force of 2,550,000 pounds.

Tendons 17 and 18 represent the eight continuity tendons in the side spans and center span respectively. These tendons are parabolically draped in the webs and their geometry is generated by the program based on the control dimensions and parameters shown in **Figure 5.2-2(d)**. Each tendon is jacked from the *left* end only with a force of 5,100,000 pounds.

Tendons 19 through 22 represent local tendons in the side span, and tendons 23 through 30 represent local tendons in the center span. These tendons are located at the bottom of the webs and are generated parametrically by the program. Tendons 19 through 24 and 29 through 30 each represent 2 actual tendons. Tendons 25 through 28 each represent four actual tendons. Tendons 19 through 22 are jacked from both ends with a force of 1,275,000 pounds. Tendons 25 through 38 are jacked from the *left* end only with a force of 2,550,000 pounds. Tendons 23, 24, 29 and 30 are jacked from the *left* end only with a force of 1,275,000 pounds.

Material properties assumed for the time dependent analysis are as follows:

The concrete, modeled using the ACI Committee 209 recommendations (Appendix B, B.2.1) has ultimate strength $f'_c(28) = 5000$ psi, ultimate creep factor $C_u(\infty) = 2.5$, ultimate shrinkage strain $\epsilon_{sh}(\infty) = 0.0008$, and unit weight $W = 155$ pcf. Time dependent

development of creep, shrinkage and aging utilize the standard ACI parameters derived for these properties with the default constants recommended by ACI. The uniformly distributed mild steel reinforcement in the frame elements has elastic modulus $E = 29,000,000$ psi. The prestressing steel has elastic modulus $E = 28,000,000$ psi., ultimate strength $f_{pu} = 270,000$ psi., prestressing steel is stress-relieved with relaxation coefficient $R = 10$, curvature friction coefficient $\mu = 0.25$ /radian, wobble coefficient $K = 0.0004$ /foot, and anchorage slip of 1/4 inch. The ultimate strength (f_{pu}) and yield stress (f_{py}) of the prestressing steel are both assumed as 270,000 psi for one solution and 210,000 for another solution. The two values are assumed the same to allow comparison between the solutions used for verification. For a regular ABI solution f_{pu} and f_{py} are entered as different values.

5.4 LOADINGS & CONSTRUCTION SEQUENCE FOR THE TIME DEPENDENT ANALYSIS

The bridge is analyzed for the actual segmental construction sequence, in which each segment is cast in place and post tensioned on a weekly schedule and the continuity tendons and superimposed dead load are added after closure, and then for a 27 year service period afterward.

The construction schedule, which is typical for this type and scale of bridge, may be summarized as follows:

Days 0 to 63:	Build pier and starting girder segments 21 through 24.
Days 63 to 168:	Build cantilever girder segments 6 through 39, stress cantilever tendons 1 through 16, with a seven day per segment cycle time.
Day 100:	Build approach girder segments 1 through 4.
Day 168:	Close side span with closure girder segment 5.
Day 175:	Stress local tendons 19 through 22 in side span.
Day 182:	Close center span with closure segment 40.
Day 189:	Stress local tendons 23 through 30 in center span, remove travelers.
Day 196:	Stress continuity tendons 17 and 18, add superimposed dead load.

A typical cycle of the segmental construction sequence is assumed to last seven days. For the cantilever construction sequence for this bridge, this results in a construction period of 105 days for the cantilever construction portion of the girder. The following steps in the analysis take place during each cycle:

1. Move traveler to location for the next segment,
2. cast segment,

3. wait three days,
4. stress the cantilever tendons anchored in the newly erected segment,
5. wait four days. For the actual bridge the four day period is used for moving the traveler and preparing the formwork for the next segment, and the three day wait is an idle period, usually scheduled to fall on a weekend.

To economize the analysis, the three day wait after casting the segment and the four day period after stressing the tendons are not separately modeled. Instead, casting of the concrete and stressing of the tendons are assumed to take place on the same day, and the concrete material parameter tables from which the elastic modulus and creep potentials are derived are generated for minimum loading age of three days. The element dead load and the prestressing are applied separately in two zero-length time steps, then a seven day time step brings the structure to the start of the next construction cycle. Thus the segmental construction analysis requires 48 solution steps.

The time dependent analysis for the 27 years (10,000) days following construction is performed in 20 time steps. Ten time steps are used between the end of construction on day 196 and day 300. Ten more time steps are used between day 300 and day 10,000. Over each of these two time intervals, the time step lengths are generated internally by the program for equal time step length on a logarithmic scale. The details of this construction sequence may be found by examination of the input for the analysis given in the following.

5.5 COMPUTER INPUT

```

=====
;   ADAPT-BRIDGE-INCREMENTAL      ABI      SOFTWARE MANUAL EXAMPLES
=====
;   Name of this file : EX5A-MNL
;           units are all in pound, inch

START

TITLE N=2
      Three span cantilever construction prestressed concrete bridge
      Adapted from reference [Ketchum, Scordelis 1986]

UNITS U=USA
CONCRETE PARAMETERS N=1 ! 1  M=ACI
MESH INPUT
  NODES N=43
    1  X=0      Y=-46.74  ! 2  X=180  Y=-40.75  ! 3  X=360  Y=-34.50
    4  X=540    Y=-34.50  ! 5  X=720  Y=-34.50  ! 6  X=788  Y=-34.61
    7  X=948    Y=-35.37  ! 8  X=1108 Y=-36.65  ! 9  X=1268 Y=-38.53
   10 X=1428    Y=-43.70  ! 11 X=1588 Y=-49.21  ! 12 X=1748 Y=-55.10
   13 X=1908    Y=-61.40  ! 14 X=2068 Y=-68.12  ! 15 X=2228 Y=-75.29
   16 X=2388    Y=-82.90  ! 17 X=2548 Y=-90.94  ! 18 X=2708 Y=-99.41
   19 X=2868    Y=-108.29 ! 20 X=3028 Y=-117.58 ! 21 X=3188 Y=-127.25
   22 X=3348    Y=-137.29 ! 23 X=3420 Y=-141.92 ! 24 X=3492 Y=-137.29
   25 X=3652    Y=-127.25 ! 26 X=3812 Y=-117.58 ! 27 X=3972 Y=-108.29
   28 X=4132    Y=-99.41  ! 29 X=4292 Y=-90.94  ! 30 X=4452 Y=-82.90
   31 X=4612    Y=-75.29  ! 32 X=4772 Y=-68.12  ! 33 X=4932 Y=-61.40
   34 X=5092    Y=-55.10  ! 35 X=5252 Y=-49.21  ! 36 X=5412 Y=-43.70
   37 X=5572    Y=-38.53  ! 38 X=5732 Y=-36.65  ! 39 X=5892 Y=-35.37
   40 X=6052    Y=-34.61  ! 41 X=6120 Y=-34.50  ! 42 X=3420 Y=-300.0
   43 X=3420    Y=-900.0

SEQUENCE
      G=1, 23, 1
      G=42, 43, 1
      G=24, 41, 1

MILD STEEL PROPERTIES N=2
      1 Es=29E6  P=0.02
      2 Es=29E6  P=0.1

CONCRETE PROPERTIES N=1
      1 Fpc=5000 Cr=3 Sh=0.0008 W=155/1728

SECTION PROPERTIES N=25
      1 B=744, 40, 349.46 D=14.375, 84.73, 9.00 ; Cant. seg.
      2 B=744, 40, 349.07 D=14.375, 85.92, 9.00
      3 B=744, 40, 348.00 D=14.375, 89.12, 9.00
      4 B=744, 40, 346.45 D=14.375, 93.77, 9.00
      5 B=744, 40, 344.70 D=14.375, 98.36, 10.34
      6 B=744, 40, 342.74 D=14.375, 103.02, 12.79

```

```

7 B=744, 40, 340.41 D=14.375, 108.77, 15.23
8 B=744, 40, 337.75 D=14.375, 115.55, 17.68
9 B=744, 40, 334.75 D=14.375, 123.31, 20.12
10 B=744, 40, 331.44 D=14.375, 132.01, 22.57
11 B=744, 40, 327.83 D=14.375, 141.63, 25.01
12 B=744, 40, 323.92 D=14.375, 152.13, 27.46
13 B=744, 40, 319.73 D=14.375, 163.48, 29.90
14 B=744, 40, 315.26 D=14.375, 175.67, 32.34
15 B=744, 40, 310.52 D=14.375, 188.68, 34.79
16 B=744, 40, 305.51 D=14.375, 202.48, 37.23
17 B=744, 40, 300.24 D=14.375, 217.06, 39.68
18 B=744, 40, 296.26 D=14.375, 228.11, 41.45 ; Pier table
19 B=744, 40, 295.00 D=14.375, 231.62, 42.00 ; Rigid zone
20 B=744, 40, 351.27 D=14.375, 74.00, 19.63 ; Near abutment
21 B=744, 40, 349.81 D=14.375, 82.75, 10.88
22 B=744, 40, 349.50 D=14.375, 84.63, 9.00
23 B=744, 40, 349.50 D=14.375, 84.63, 9.00
24 B=348 D=96 ; Pier
25 B=72,1.5,72 D=4,80,4 ; Traveler

```

ELEMENTS N=42

FRAME N=42

```

1,1,2 C=1 X=20 St=1 Day=100 G=1,4,1,1,1,0,1,0,0
5,5,6 C=1 X=1 St=1 Day=168 G=5,22,1,1,1,0,1,0,-7
23,23,24 C=1 X=18 St=1 Day=49 G=23,40,1,1,1,0,-1,0,7
41,23,42 C=1 X=19 St=2 Day=0
42,42,43 C=1 X=24 St=2 Day=0

```

PRESTRESSING STEEL N=1

1 Ep=28E6 Meu=.25 K=.0004/12 Fpu=27E4 R=10

TENDON GEOMETRY N=30

```

; .....Cantiver tendons
; each represents 4, 21-Strand, 1/2" Diam tendons
1 Spans=2 M=1 Area=12.852
1 N=3 G=21,23,1
R=0, .6638,0 S=-20,-7,-7
2 N=3 G=23,25,1
R=0, .3362,0 S=-7,-7,-20
2 Spans=2 M=1 Area=12.852
1 N=4 G=20,23,1
R=0, .3929,0 S=-20,-7,-7
2 N=4 G=23,26,1
R=0, .6071,0 S=-7,-7,-20
3 Spans=2 M=1 Area=12.852
1 N=5 G=19,23,1
R=0, .2790,0 S=-20,-7,-7
2 N=5 G=23,27,1
R=0, .7210,0 S=-7,-7,-20
4 Spans=2 M=1 Area=12.852
1 N=6 G=18,23,1
R=0, .2163,0 S=-20,-7,-7
2 N=6 G=23,28,1
R=0, .7837,0 S=-7,-7,-20

```

- 5 Spans=2 M=1 Area=12.852
 1 N=7 G=17,23,1
 R=0, .1766,0 S=-20,-7,-7
 2 N=7 G=23,29,1
 R=0, .8234,0 S=-7,-7,-20
- 6 Spans=2 M=1 Area=12.852
 1 N=8 G=16,23,1
 R=0, .1492,0 S=-20,-7,-7
 2 N=8 G=23,30,1
 R=0, .8508,0 S=-7,-7,-20
- 7 Spans=2 M=1 Area=12.852
 1 N=9 G=15,23,1
 R=0, .1292,0 S=-20,-7,-7
 2 N=9 G=23,31,1
 R=0, .8708,0 S=-7,-7,-20
- 8 Spans=2 M=1 Area=12.852
 1 N=10 G=14,23,1
 R=0, .1139,0 S=-20,-7,-7
 2 N=10 G=23,32,1
 R=0, .8861,0 S=-7,-7,-20
- 9 Spans=2 M=1 Area=12.852
 1 N=11 G=13,23,1
 R=0, .1019,0 S=-20,-7,-7
 2 N=11 G=23,33,1
 R=0, .8981,0 S=-7,-7,-20
- 10 Spans=2 M=1 Area=12.852
 1 N=12 G=12,23,1
 R=0, .0921,0 S=-20,-7,-7
 2 N=12 G=23,34,1
 R=0, .9079,0 S=-7,-7,-20
- 11 Spans=2 M=1 Area=12.852
 1 N=13 G=11,23,1
 R=0, .0841,0 S=-20,-7,-7
 2 N=13 G=23,35,1
 R=0, .9159,0 S=-7,-7,-20
- 12 Spans=2 M=1 Area=12.852
 1 N=14 G=10,23,1
 R=0, .0773,0 S=-20,-7,-7
 2 N=14 G=23,36,1
 R=0, .9277,0 S=-7,-7,-20
- 13 Spans=2 M=1 Area=12.852
 1 N=15 G=9,23,1
 R=0, .0716,0 S=-20,-7,-7
 2 N=15 G=23,37,1
 R=0, .9284,0 S=-7,-7,-20
- 14 Spans=2 M=1 Area=12.852
 1 N=16 G=8,23,1
 R=0, .0666,0 S=-20,-7,-7
 2 N=16 G=23,38,1
 R=0, .9334,0 S=-7,-7,-20
- 15 Spans=2 M=1 Area=12.852
 1 N=17 G=7,23,1
 R=0, .0623,0 S=-20,-7,-7
 2 N=17 G=23,39,1

```

        R=0, .9377,0 S=-7,-7,-20
16 Spans=2 M=1 Area=12.852
    1 N=18 G=6,23,1
        R=0, .0585,0 S=-20,-7,-7
    2 N=18 G=23,40,1
        R=0, .9415,0 S=-7,-7,-20
; .....Continuity tendons
;     each represents 8, 21-Strand, 1/2" Diam tendons
17 Spans=2 M=1 Area=25.704
    1 N=18 G=1,18,1
        R=0, .4,.2 S=-46.74,-100,-12
    2 N=6 G=18,23,1
        R=0, .5, 0 S=-12,-12,-12
18 Spans=2 M=1 Area=25.704
    1 N=6 G=23,28,1
        R=0, .5, 0 S=-12,-12,-12
    2 N=14 G=28,41,1
        R=.3, 1, 0 S=-12,-100,-100
; .....Local tendons
;     each represents 2 or 4, 21-strand, 1/2" Diam tendons
19 Spans=1 M=1 Area=6.426
    1 N=9 G=1,9,1
        R=0, .874, 0 S=-46.74,-98.15,-103.34
20 Spans=1 M=1 Area=6.426
    1 N=10 G=1,10,1
        R=0, .776, 0 S=-46.74,-98.15,-107.43
21 Spans=1 M=1 Area=6.426
    1 N=11 G=1,11,1
        R=0, .698, 0 S=-46.74,-98.15,-112.64
22 Spans=1 M=1 Area=6.426
    1 N=12 G=1,12,1
        R=0, .634, 0 S=-46.74,-98.15,-118.91
23 Spans=1 M=1 Area=6.426
    1 N=12 G=30,41,1
        R=0, 1, 0 S=-153.64,-100,-100
24 Spans=1 M=1 Area=6.426
    1 N=11 G=31,41,1
        R=0, 1, 0 S=-143.58,-100,-100
25 Spans=1 M=1 Area=12.852
    1 N=10 G=32,41,1
        R=0, 1, 0 S=-134.42,-100,-100
26 Spans=1 M=1 Area=12.852
    1 N=9 G=33,41,1
        R=0, 1, 0 S=-126.18,-100,-100
27 Spans=1 M=1 Area=12.852
    1 N=8 G=34,41,1
        R=0, 1, 0 S=-118.91,-100,-100
28 Spans=1 M=1 Area=12.852
    1 N=7 G=35,41,1
        R=0, 1, 0 S=-112.64,-100,-100
29 Spans=1 M=1 Area=6.426
    1 N=6 G=36,41,1
        R=0, 1, 0 S=-107.43,-100,-100
30 Spans=1 M=1 Area=6.426

```

```

1 N=5 G=37,41,1
  R=0, 1, 0 S=-103.34, -100, -100

TRAVELERS N=3
  1 X=25 E=29E6 W=15E4 N=3
  2                               N=3
  3 X=25 E=29E6 W=1E4 N=2

MESH COMPLETE ! SET Day=56

; .....Build pier and starting segment

CHANGE STRUCTURE
  RESTRAINTS
    1,5 R=0,1,0
    41 R=1,0,1
    43 R=1,1,1

    BUILD N=41,42
    BUILD N=21,24 Day=35
CHANGE COMPLETE
SOLVE ! SOLVE A=0 Day=63 ! OUTPUT

; .....Stress tendon 1, build travelers, build segments 20 and 25
CHANGE STRUCTURE
  STRESS N=1 StressTo=198E3,198E3 Anchor=.25,.25
  MOVE N=1 D=20,21,22
  MOVE N=2 D=24,25,26
CHANGE COMPLETE
SOLVE ! OUTPUT

CHANGE STRUCTURE
  BUILD N=20,25,5
CHANGE COMPLETE
SOLVE !SOLVE Day=70

; .....Stress tendon 2, move travelers, build segments 19 and 26
CHANGE STRUCTURE
  STRESS N=2 StressTo=198E3,198E3 Anchor=.25,.25
  MOVE N=1 D=19,20,21
  MOVE N=2 D=25,26,27
CHANGE COMPLETE
SOLVE !OUTPUT

CHANGE STRUCTURE
  BUILD N=19,26,7
CHANGE COMPLETE
SOLVE !SOLVE Day=77

; .....Stress tendon 3, move travelers, build segments 18 and 27
CHANGE STRUCTURE
  STRESS N=3 StressTo=198E3,198E3 Anchor=.25,.25
  MOVE N=1 D=18,19,20
  MOVE N=2 D=26,27,28

```

CHANGE COMPLETE !SOLVE !OUTPUT

CHANGE STRUCTURE
BUILD N=18,27,9
CHANGE COMPLETE
SOLVE !SOLVE Day=84

; Stress tendon 4, move travelers, build segments 17 and 28
CHANGE STRUCTURE
STRESS N=4 StressTo=198E3,198E3 Anchor=.25,.25
MOVE N=1 D=17,18,19
MOVE N=2 D=27,28,29
CHANGE COMPLETE
SOLVE !OUTPUT

CHANGE STRUCTURE
BUILD N=17,28,11
CHANGE COMPLETE
SOLVE !SOLVE Day=91

; Stress tendon 5, move travelers, build segments 16 and 29
CHANGE STRUCTURE
STRESS N=5 StressTo=198E3,198E3 Anchor=.25,.25
MOVE N=1 D=16,17,18
MOVE N=2 D=28,29,30
CHANGE COMPLETE
SOLVE !OUTPUT

CHANGE STRUCTURE
BUILD N=16,29,13
CHANGE COMPLETE
SOLVE !SOLVE Day=98

; Stress tendon 6, move travelers, build segments 15 and 30
CHANGE STRUCTURE
STRESS N=6 StressTo=198E3,198E3 Anchor=.25,.25
MOVE N=1 D=15,16,17
MOVE N=2 D=29,30,31
CHANGE COMPLETE
SOLVE !OUTPUT

CHANGE STRUCTURE
BUILD N=15,30,15
CHANGE COMPLETE
SOLVE !SOLVE Day=105

; Stress tendon 7, move travelers, build segments 14 and 31
CHANGE STRUCTURE
STRESS N=7 StressTo=198E3,198E3 Anchor=.25,.25
MOVE N=1 D=14,15,16
MOVE N=2 D=30,31,32
CHANGE COMPLETE
SOLVE !OUTPUT

```
CHANGE STRUCTURE
      BUILD N=14,31,17
CHANGE COMPLETE
SOLVE !SOLVE Day=112

; .....Stress tendon 8, move travelers, build segmente 13 and 32
CHANGE STRUCTURE
      STRESS N=8 StressTo=198E3,198E3 Anchor=.25,.25
      MOVE N=1 D=13,14,15
      MOVE N=2 D=31,32,33
CHANGE COMPLETE
SOLVE !OUTPUT

CHANGE STRUCTURE
      BUILD N=13,32,19
CHANGE COMPLETE
SOLVE !SOLVE Day=119

; .....Stress tendon 9, move travelers, build segments 12 and 33
CHANGE STRUCTURE
      STRESS N=9 StressTo=198E3,198E3 Anchor=.25,.25
      MOVE N=1 D=12,13,14
      MOVE N=2 D=32,33,34
CHANGE COMPLETE
SOLVE !OUTPUT

CHANGE STRUCTURE
      BUILD N=12,33,21
CHANGE COMPLETE
SOLVE !SOLVE Day=126

; .....Stress tendon 10, move travelers, build segments 11 and 34
CHANGE STRUCTURE
      STRESS N=10 StressTo=198E3,198E3 Anchor=.25,.25
      MOVE N=1 D=11,12,13
      MOVE N=2 D=33,34,35
CHANGE COMPLETE
SOLVE !OUTPUT

CHANGE STRUCTURE
      BUILD N=11,34,23
CHANGE COMPLETE
SOLVE !SOLVE Day=133

; .....Stress tendon 11, move travelers, build segments 10 and 35
CHANGE STRUCTURE
      STRESS N=11 StressTo=198E3,198E3 Anchor=.25,.25
      MOVE N=1 D=10,11,12
      MOVE N=2 D=34,35,36
CHANGE COMPLETE
SOLVE !OUTPUT

CHANGE STRUCTURE
      BUILD N=10,35,25
```

CHANGE COMPLETE
SOLVE !SOLVE Day=140

; Stress tendon 12, move travelers, build segments 9 and 36

CHANGE STRUCTURE
 STRESS N=12 StressTo=198E3,198E3 Anchor=.25,.25
 MOVE N=1 D=9,10,11
 MOVE N=2 D=35,36,37

CHANGE COMPLETE
SOLVE !OUTPUT

CHANGE STRUCTURE
 BUILD N=9,36,27

CHANGE COMPLETE
SOLVE !SOLVE Day=147

; Stress tendon 13, move travelers, build segments 8 and 37

CHANGE STRUCTURE
 STRESS N=13 StressTo=198E3,198E3 Anchor=.25,.25
 MOVE N=1 D=8,9,10
 MOVE N=2 D=36,37,38

CHANGE COMPLETE
SOLVE !OUTPUT

CHANGE STRUCTURE
 BUILD N=8,37,29

CHANGE COMPLETE
SOLVE !SOLVE Day=154

; Stress tendon 14, move travelers, build segments 7 and 38

CHANGE STRUCTURE
 STRESS N=14 StressTo=198E3,198E3 Anchor=.25,.25
 MOVE N=1 D=7,8,9
 MOVE N=2 D=37,38,39

CHANGE COMPLETE
SOLVE !OUTPUT

CHANGE STRUCTURE
 BUILD N=7,38,31

CHANGE COMPLETE
SOLVE !SOLVE Day=161

; Stress tendon 15, move travelers, build segments 6 and 39

CHANGE STRUCTURE
 STRESS N=15 StressTo=198E3,198E3 Anchor=.25,.25
 MOVE N=1 D=6,7,8
 MOVE N=2 D=38,39,40

CHANGE COMPLETE
SOLVE !OUTPUT

CHANGE STRUCTURE
 BUILD N=6,39,33

CHANGE COMPLETE
SOLVE !SOLVE Day=168

```
; .....Stress tendon 16
CHANGE STRUCTURE
    STRESS N=16 StressTo=198E3,198E3 Anchor=.25,.25
CHANGE COMPLETE
SOLVE A=0.71 !OUTPUT

; .....Build segments 1 thru 4, install closure formwork,
;           build closure segment 5
CHANGE STRUCTURE
    BUILD N=1,4
    MOVE  N=3 D=5,6
    BUILD N=5
CHANGE COMPLETE
SOLVE !SOLVE Day=175 Steps=3

; .....Remove supports at nodes 2 thru 5, stress tendons 19 thru 22
CHANGE STRUCTURE
    RESTRAINTS
        2,5 R=0,0,0

    STRESS N=19,22 StressTo=198E3,198E3 Anchor=.25,.25
CHANGE COMPLETE
SOLVE !SOLVE Day=182 Steps=3

; .....Move closure formwork to center span, build closure segment 40
CHANGE STRUCTURE
    MOVE  N=3 D=40,41
    BUILD N=40 Day=182
CHANGE COMPLETE
SOLVE !SOLVE Day=189 Steps=3

; .....Stress tendons 23 thru 30, remove travelers,
;           and closure formwork
CHANGE STRUCTURE
    STRESS N=23,30 StressTo=198E3 Anchor=.25
    MOVE  N=1 D=0
    MOVE  N=2 D=0
    MOVE  N=3 D=0
CHANGE COMPLETE
SOLVE !SOLVE Day=196 Steps=3

; .....Stress continuity tendons, add remaining dead load
CHANGE STRUCTURE
    STRESS N=17,18 StressTo=198E3 Anchor=.25
CHANGE COMPLETE

LOADING
    L=1,40 F=0,-2500/12

SOLVE !OUTPUT

; .....Step through time up to 10000 days (27.4 years)
```

```
SOLVE Day=300 Steps=10
SOLVE Day=10000 Steps=10 !OUTPUT
STOP
```

5.6 COMPARISON OF RESULTS

Using ABI, two sets of solutions are obtained for the bridge example. The solutions are compared with those obtained using SFRAME and SPCFRAME [Scordelis 1993]. One solution is for the prestressing yield stress (f_{py}) of 270 ksi, the second for a yield stress of 210 ksi. The yield stress is defined as stress at 0.1% offset from the stress-strain curve of the prestressing material. The values selected for the yield stress are those used in the reference analyses. Observe that in the input data given in Section 5.5 the ultimate stress (f_{pu}) and the yield stress (f_{py}) are entered with the same value. In a regular ABI analysis these values would be different.

The solutions obtained are listed with those of reference [Scordelis, 1993] in Table 5.6-1. The agreement among the solutions is very close. All values agree within 1%, except for deflection at line of symmetry, where ABI deflections are larger than Sframe by 11%. Moments, hence stresses, tendon stresses and the reactions are, from an engineering standpoint, almost identical with Sframe solutions.

TABLE 5.6-1 KETCHUM BRIDGE-COMPARISON OF ANALYSIS RESULTS

	f_{py} (ksi)	After Construction				After 27.4 Years			
		SFRAME	SPCFRAME	ADAPT-	SFRAME	SPCFRAME	ADAPT-ABI		
Central Deflection	270 210	4.38 5.70	4.20 5.48	4.98 6.42	9.09 11.46	8.64 10.81	10.44 12.79		
Reaction at Abutment	270 210	1.25E+06 1.25E+06	1.25E+06 1.24E+06	1.25E+06 1.25E+06	1.25E+06 1.28E+06	1.25E+06 1.28E+06	1.26E+06 1.29E+06		
Reaction at Pier	270 210	1.48E+07 1.48E+07	1.48E+07 1.48E+07	1.48E+07 1.48E+07	1.48E+07 1.48E+07	1.48E+07 1.48E+07	1.48E+07 1.47E+07		
Moment at Pier	270 210	7.29E+09 7.28E+09	7.30E+09 7.32E+09	7.29E+09 7.28E+09	7.28E+09 7.20E+09	7.29E+09 7.20E+09	7.24E+09 7.16E+09		
Moment at Center	270 210	9.65E+08 9.39E+08	9.69E+08 9.39E+08	9.65E+08 9.39E+08	1.05E+09 1.01E+09	1.10E+09 1.04E+09	1.04E+09 1.00E+09		
Tendon Stress at Pier	270 210	1.64E+05 1.64E+05	1.64E+05 1.64E+05	1.64E+05 1.64E+05	1.53E+05 1.40E+05	1.53E+05 1.39E+05	1.53E+05 1.40E+05		
Tendon Stress at Center (psi)	270 210	1.74E+05 1.74E+05	1.74E+05 1.74E+05	1.74E+05 1.74E+05	1.43E+05 1.31E+05	1.41E+05 1.30E+05	1.41E+05 1.30E+05		

Note: f_{py} is assumed 0.1% offset yield stress for calculation of prestress relaxation only.

5.7 AMERICAN, SI AND MKS SYSTEM OF UNITS EXAMPLE

This example is devised to verify the correlation accuracy of the software among its built-in three different systems of units *American*, *SI*, and *MKS*. It achieves two objectives. First, by providing a complete listing of input data for each of the system of units implemented, it demonstrates the specific features of data preparation for each. Second, it substantiates the validity of the software in changing from one system of units to the next. It confirms that solutions to the same problem, obtained in different systems of units are practically identical.

To obtain a correct solution, it is imperative that units entered for each system be consistent with the listing given below.

AMERICAN	:	pound,	inch,	degrees F
SI	:	Newtons,	mm,	degree C
MKS	:	Kg,	cm,	degree C

The conversion factors used in the program for the more frequent items are listed in Table 5.7-1.

TABLE 5.7-1 PRINCIPAL CONVERSION UNITS USED IN ABI

	American	SI	MKS
Length	1 inch	2.54000E+01 mm	2.54000E+00 cm
Area	1 inch ²	6.45160E+02 mm ²	6.45160E+00 cm ²
Moment of inertia	1 inch ⁴	4.16231E+05 mm ⁴	4.16231E+01 cm ⁴
Weight	1 lb	4.53438E+00 kg	4.53438E-01 kg
Force	1 lb	4.44822E+00 N	4.53438E-01 kg
Moment	1 lb-in.	1.12985E+02 N-mm	1.15173E+00 kg-cm
Stress	1 psi	6.89476E+02 MPa	7.02830E-02 kg-cm ²
Temperature	1 °F	5.55556E-01 °C	5.55556E-01 °C

The structural model selected for verification is chosen to be simple, yet contain the dimensionally critical features. The model is not meant to represent a prototype.

5.8 STRUCTURE

The model (refer to **Figure 5.8-1**) is a post-tensioned cantilever resting on a spring at its mid-length. It is subject to self-weight, applied concentrated loading, temperature, creep,

shrinkage and aging of concrete. Details of the model are listed in Table 5.8-1. Six conditions, consisting of two concrete material models (ACI, CEB1), each with three systems of units (American, SI and MKS) are investigated. The model's response to loading and the time-dependent effects are computed at days 10, 500 and 1000. The actual input files are appended to this description.

TABLE 5.8-1 DIMENSIONS, PROPERTIES AND LOADING OF THE VERIFICATION MODEL

Item	Symbol	USA lb; inch; ° F	SI N; mm; ° C	MKS Kg; cm; ° C
Length	L	120	3.0480E+03	3.0480E+02
Area	A	300	1.9355E+05	1.9355E+03
Moment of Inertia	I	2200	9.1571E+08	9.1571E+04
Top fiber	C _{top}	11	2.7940E+02	2.7940E+01
Bottom fiber	C _{bot}	17	4.3180E+02	4.3180E+01
Concrete strength	f _c	5000	3.4486E+01	3.5154E+02
Unit weight	W	8.6806E-02 lb/in ³	2.4019E-06 kg/mm ³	2.4019E-03 kg/cm ³
Thermal coef.	α	5.5000E-06	9.9000E-06	9.9000E-06
Mild steel	E _s	2.9000E+07	2.0002E+05	2.0389E+06
	α	6.0000E-06	10.800E-06	10.800E-06
Prestressing values	E _{ps}	2.8000E+07	1.9312E+05	1.9686E+06
	α	5.0000E-06	9.0000E-06	9.0000E-06
	C - Curvature	0.25/radian	0.25/radian	0.25/radian
	W - Wobble	3.0000E-04	1.1811E-05	1.1811E-04
Spring stiffness	K	3.0000E+04 lb/in	5.2555E+03 N/mm	5.3575E+03 kg/cm
Load values	P _v	-70	-3.1148E+02	-3.1752E+01
	P _H	-35000	-1.5574E+05	-1.5875E+04
Temperature	Ambient	68	20	20
	Solution	158	70	70

5.9 RESULTS

The solution at day 1000 for stress and deflection at node 2 of the verification model, together with total top fiber concrete stress at node I of element 2 are listed in Table 5.9-1 for comparison. For ease of reference, the values obtained for SI and MKS units are duplicated in their equivalents in the American system of units (in

parenthesis) below the computer output of the original units. Solutions of each material model (ACI and CEB1) relate together.

Note that the agreement among the three systems of units is very good.

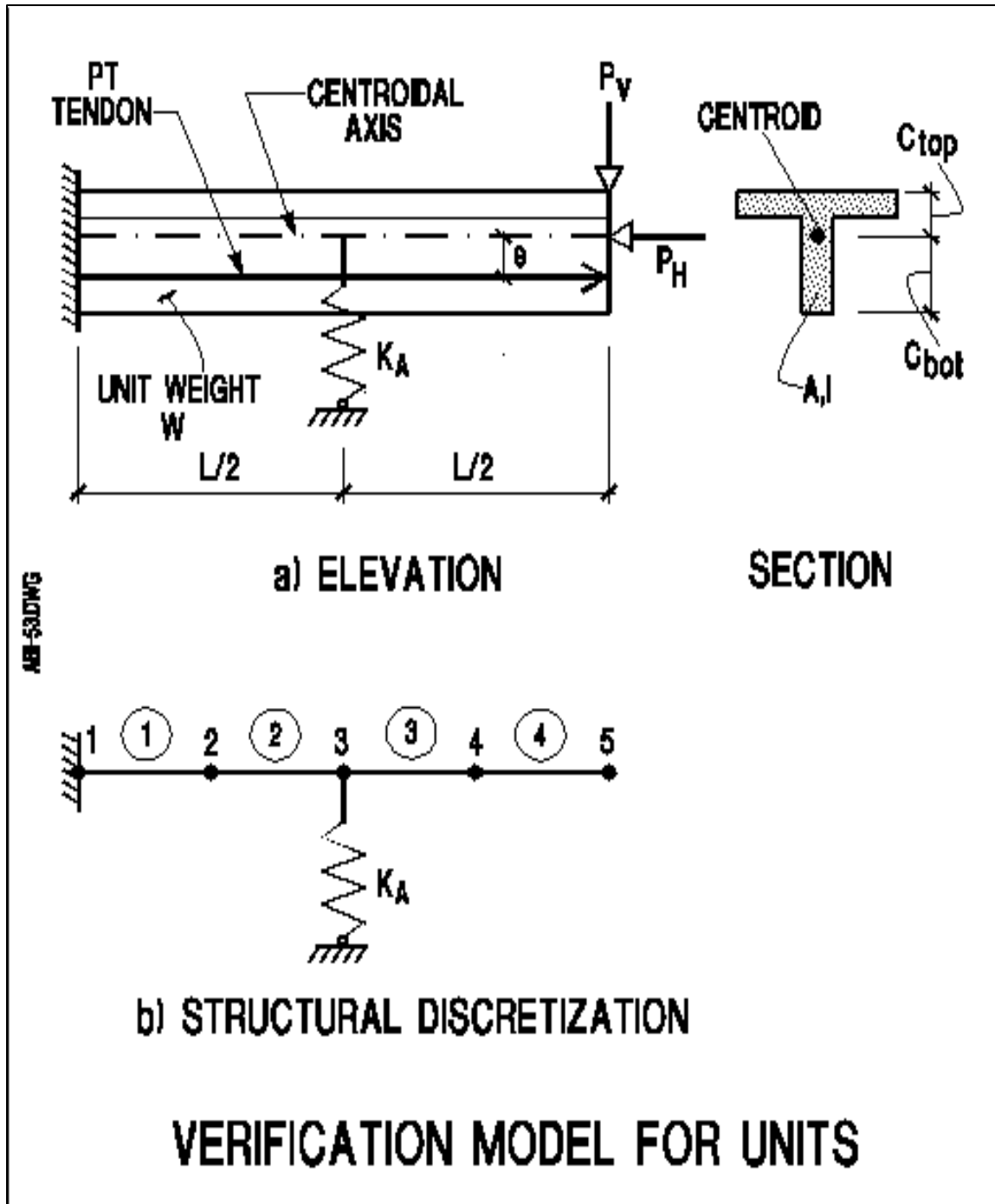


FIGURE 5.8-1

TABLE 5.9-1 COMPARISON OF SOLUTIONS OBTAINED WITH DIFFERENT UNITS

	AMERICAN		SI		MKS	
	Deflection in	Stress psi	Deflection mm (in.)	Stress MPa (psi)	Deflection cm (in.)	Stress kg/cm ² (psi)
ACI	1.5934E-02	1.3944E+03	4.0473E-01 (1.5934E-02)	9.6176E+00 (1.395E+03)	4.0473E-02 (1.5934E-03)	9.8039E+01 (1.394E+03)
CEB1	1.7428E-02	1.3433E+03	4.4267E-01 (1.7428E-02)	9.2647E+00 (1.344E+03)	4.4267E-02 (1.7428E-03)	9.4441E+01 (1.343E+03)

```

;=====
;   ADAPT-BRIDGE-INCREMENTAL   ABI   SOFTWARE MANUAL EXAMPLES
;=====
; name of this file: UACI

START
TITLE N=1
      ACI AMERICAN UNITS      lb-inch
UNITS U=USA

CONCRETE PARAMETERS N=1
1 M=ACI  A=4 B=0.85 C=1.25 D=0.118 E=1 F=50 W=0.086805556

MESH INPUT
  NODES N=6
    1 X=0      Y=0
    5 X=120    Y=0 G=1,5
    6 X=60     Y=-25

  CONCRETE PROPERTIES N=1
    1 Fpc=5000 Cr=2.5 Sh=0.0003 W=0.086805556 Ac=0.0000055
  MILD STEEL PROPERTIES N=1
    1 Es=29000000.0 P=0.02 As=0.000006
  SECTION PROPERTIES N=1
    1 Area=300 I=2200 C=11,17
  ELEMENTS N=5
    FRAME N=4
      1,1,2 C=1 X=1 St=1 Day=1 G=1,4,1,1,1
    SPRINGS N=1
      5,3,6 K=30000

  PRESTRESSING STEEL N=1
    1 Ep=28000000 Meu=0.25 K=0.0003 Fpu=270000 R=45 Ap=0.000005
  TENDON GEOMETRY N=1
    1 Spans=1 M=1 Area=0.153
      1 N=5 G=1,5,1 B=0,0 E=10,0
        R=0,0.5,0 S=-2,-2,-2

MESH COMPLETE

SET Day=10 T=68
CHANGE STRUCTURE
  BUILD N=1,5,1
  STRESS N=1 StressTo=0,200E3 Anchor=0,0.125
  RESTRAINTS
    1 R=1,1,1
    6 R=1,1,1

CHANGE COMPLETE
LOADING
  N=5 F=-35000,-70,0

SOLVE          ! OUTPUT
SOLVE Day=500 Steps=3 ! OUTPUT
LOADING
L=1,4,1 T=158,158

SOLVE Day=1000 Steps=3 ! OUTPUT
STOP

```

```

;=====
;   ADAPT-BRIDGE-INCREMENTAL   ABI   SOFTWARE MANUAL EXAMPLE
;=====
; name of this file: SACI

START
TITLE N=1
      ACI SI UNITS      N-mm
UNITS U=SI

CONCRETE PARAMETERS N=1
1 M=ACI  A=4 B=0.85 C=1.25 D=0.118 E=1 F=50 W=0.0000024027941

MESH INPUT
  NODES N=6
    1 X=0      Y=0
    5 X=3048   Y=0 G=1,5
    6 X=1524   Y=-635

  CONCRETE PROPERTIES N=1
    1 Fpc=34.4859 Cr=2.5 Sh=0.0003 W=0.0000024027941 Ac=0.0000099
  MILD STEEL PROPERTIES N=1
    1 Es=200018.32 P=0.02 As=0.0000108
  SECTION PROPERTIES N=1
    1 Area=193548 I=9.1570913E8 C=279.4,431.8
  ELEMENTS N=5
    FRAME N=4
      1,1,2 C=1 X=1 St=1 Day=1 G=1,4,1,1,1
    SPRINGS N=1
      5,3,6 K=5255.65

  PRESTRESSING STEEL N=1
    1 Ep=193121 Meu=0.25 K=0.000011811 Fpu=1862.24 R=45 Ap=0.000009
  TENDON GEOMETRY N=1
    1 Spans=1 M=1 Area=98.70948
      1 N=5 G=1,5,1 B=0,0 E=254,0
        R=0,0.5,0 S=-50.8, -50.8, -50.8

MESH COMPLETE

SET Day=10 T=20

CHANGE STRUCTURE
  BUILD N=1,5,1
  STRESS N=1 StressTo=0,1379.437 Anchor=0,3.175
  RESTRAINTS
    1 R=1,1,1
    6 R=1,1,1

CHANGE COMPLETE

LOADING
  N=5 F=-155742.5, -311.485,0

SOLVE      ! OUTPUT
SOLVE Day=500 Steps=3 ! OUTPUT
LOADING
  L=1,4,1 T=70,70

SOLVE Day=1000 Steps=3 ! OUTPUT
STOP

```

```

;=====
;   ADAPT-BRIDGE-INCREMENTAL   ABI   SOFTWARE MANUAL EXAMPLES
;=====
;   name of this file: MACI

START
TITLE N=1
      ACI MKS UNITS      kg-cm
UNITS U=MKS

CONCRETE PARAMETERS N=1
1 M=ACI  A=4 B=0.85 C=1.25 D=0.118 E=1 F=50 W=0.0024027941

MESH INPUT
  NODES N=6
    1 X=0      Y=0
    5 X=304.8   Y=0 G=1,5
    6 X=152.4   Y=-63.5

  CONCRETE PROPERTIES N=1
    1 Fpc=351.538 Cr=2.5 Sh=0.0003 W=0.0024027941 Ac=0.0000099
  MILD STEEL PROPERTIES N=1
    1 Es=2.038923E6 P=0.02 As=0.0000108
  SECTION PROPERTIES N=1
    1 Area=1935.48 I=9.1570913E4 C=27.94,43.18
  ELEMENTS N=5
    FRAME N=4
      1,1,2 C=1 X=1 St=1 Day=1 G=1,4,1,1,1
    SPRINGS N=1
      5,3,6 K=5357.45

  PRESTRESSING STEEL N=1
    1 Ep=1.968615E6 Meu=0.25 K=0.00011811 Fpu=18983.07 R=45 Ap=0.000009
  TENDON GEOMETRY N=1
    1 Spans=1 M=1 Area=0.9870948
      1 N=5 G=1,5,1 B=0,0 E=25.4,0
        R=0,0.5,0 S=-5.08,-5.08,-5.08

MESH COMPLETE

SET Day=10 T=20

CHANGE STRUCTURE
  BUILD N=1,5,1
  STRESS N=1 StressTo=0,14061.536 Anchor=0,0.3175
  RESTRAINTS
    1 R=1,1,1
    6 R=1,1,1

CHANGE COMPLETE
LOADING
  N=5 F=-15875.9,-31.752,0

SOLVE      ! OUTPUT
SOLVE Day=500 Steps=3 ! OUTPUT
LOADING
  L=1,4,1 T=70,70
SOLVE

SOLVE Day=1000 Steps=3 ! OUTPUT
STOP

```

```

;=====
;   ADAPT-BRIDGE-INCREMENTAL   ABI   SOFTWARE MANUAL EXAMPLE
;=====
; name of this file: UCEB1

START
TITLE N=1
      CEB1 AMERICAN UNITS      lb-inch
UNITS U=USA

CONCRETE PARAMETERS N=1
1 M=CEB1      Area=300      P=80

MESH INPUT
  NODES N=6
    1 X=0      Y=0
    5 X=120    Y=0      G=1,5
    6 X=60     Y=-25

  CONCRETE PROPERTIES N=1
    1 Fpc=5000 Cr=2.5 Sh=0.0003      W=0.086805556      Ac=0.0000055
  MILD STEEL PROPERTIES N=1
    1 Es=29000000.0 P=0.02      As=0.000006
  SECTION PROPERTIES N=1
    1 Area=300 I=2200 C=11,17
  ELEMENTS N=5
    FRAME N=4
      1,1,2 C=1 X=1 St=1 Day=1 G=1,4,1,1,1
    SPRINGS N=1
      5,3,6 K=30000

  PRESTRESSING STEEL N=1
    1 Ep=28000000 Meu=0.25 K=0.0003 Fpu=270000 R=45 Ap=0.000005
  TENDON GEOMETRY N=1
    1 Spans=1 M=1 Area=0.153
      1 N=5 G=1,5,1 B=0,0 E=10,0
        R=0,0.5,0 S=-2,-2,-2

MESH COMPLETE

SET Day=10      T=68

CHANGE STRUCTURE
  BUILD N=1,5,1
  STRESS N=1 StressTo=0,200E3 Anchor=0,0.125
  RESTRAINTS
    1 R=1,1,1
    6 R=1,1,1

CHANGE COMPLETE
LOADING
  N=5 F=-35000,-70,0

SOLVE      ! OUTPUT
SOLVE Day=500 Steps=3 ! OUTPUT
LOADING
  L=1,4,1 T=158,158
SOLVE

SOLVE Day=1000 Steps=3 ! OUTPUT
STOP

```

```

;=====
;   ADAPT-BRIDGE-INCREMENTAL   ABI   SOFTWARE MANUAL EXAMPLES
;=====
; name of this file: SCEB1

START
TITLE N=1
      CEB1 SI UNITS      N-mm
UNITS U=SI

CONCRETE PARAMETERS N=1
1 M=CEB1      Area=193548      P=2032

MESH INPUT
  NODES N=6
    1 X=0      Y=0
    5 X=3048    Y=0 G=1,5
    6 X=1524    Y=-635

CONCRETE PROPERTIES N=1
  1 Fpc=34.4859 Cr=2.5 Sh=0.0003 W=0.0000024027941 Ac=0.0000099
MILD STEEL PROPERTIES N=1
  1 Es=200018.32 P=0.02 As=0.0000108
SECTION PROPERTIES N=1
  1 Area=193548 I=9.1570913E8 C=279.4,431.8
ELEMENTS N=5
  FRAME N=4
    1,1,2 C=1 X=1 St=1 Day=1 G=1,4,1,1,1
  SPRINGS N=1
    5,3,6 K=5255.65

PRESTRESSING STEEL N=1
  1 Ep=193121 Meu=0.25 K=0.000011811 Fpu=1862.24 R=45 Ap=0.000009
TENDON GEOMETRY N=1
  1 Spans=1 M=1 Area=98.70948
    1 N=5 G=1,5,1 B=0,0 E=254,0
      R=0,0.5,0 S=-50.8, -50.8, -50.8

MESH COMPLETE

SET Day=10 T=20
CHANGE STRUCTURE
  BUILD N=1,5,1
  STRESS N=1 StressTo=0,1379.437 Anchor=0,3.175
  RESTRAINTS
    1 R=1,1,1
    6 R=1,1,1

CHANGE COMPLETE
LOADING
  N=5 F=-155742.5, -311.485,0

SOLVE          ! OUTPUT
SOLVE Day=500 Steps=3 ! OUTPUT
LOADING
  L=1,4,1 T=70,70
SOLVE

SOLVE Day=1000 Steps=3 ! OUTPUT
STOP

```

```

;=====
;   ADAPT-BRIDGE-INCREMENTAL   ABI   SOFTWARE MANUAL EXAMPLES
;=====
; name of this file: MCEB1

START
TITLE N=1
      CEB1 MKS UNITS      kg-cm
UNITS U=MKS

CONCRETE PARAMETERS N=1
1 M=CEB1      Area=1935.48      P=203.20

MESH INPUT
  NODES N=6
    1 X=0 Y=0
    5 X=304.8 Y=0 G=1,5
    6 X=152.4 Y=-63.5

  CONCRETE PROPERTIES N=1
    1 Fpc=351.538 Cr=2.5 Sh=0.0003 W=0.0024027941 Ac=0.0000099
  MILD STEEL PROPERTIES N=1
    1 Es=2.038923E6 P=0.02 As=0.0000108
  SECTION PROPERTIES N=1
    1 Area=1935.48 I=9.1570913E4 C=27.94,43.18
  ELEMENTS N=5
    FRAME N=4
      1,1,2 C=1 X=1 St=1 Day=1 G=1,4,1,1,1
    SPRINGS N=1
      5,3,6 K=5357.45

  PRESTRESSING STEEL N=1
    1 Ep=1.968615E6 Meu=0.25 K=0.00011811 Fpu=18983.07 R=45 Ap=0.000009
  TENDON GEOMETRY N=1
    1 Spans=1 M=1 Area=0.9870948
      1 N=5 G=1,5,1 B=0,0 E=25.4,0
        R=0,0.5,0 S=-5.08,-5.08,-5.08

MESH COMPLETE

SET Day=10 T=20
CHANGE STRUCTURE
  BUILD N=1,5,1
  STRESS N=1 StressTo=0,14061.536 Anchor=0,0.3175
  RESTRAINTS
    1 R=1,1,1
    6 R=1,1,1

CHANGE COMPLETE
LOADING
  N=5 F=-15875.9,-31.752,0

SOLVE          ! OUTPUT
SOLVE Day=500 Steps=3 ! OUTPUT
LOADING
  L=1,4,1 T=70,70
SOLVE

SOLVE Day=1000 Steps=3 ! OUTPUT
STOP

```

A. INPUT / OUTPUT EXAMPLES

A.1 OUTPUT ORGANIZATION 3
 A.1.1 Output Overview.....3
A.2 EXAMPLE 11
A.3 INPUT FILE.....12
A.4 OUTPUT LISTING 13
A.5 GRAPHICAL DISPLAY OF OUTPUT.....19
A.6 INPUT/OUTPUT GRAPHICAL DISPLAY 21

This Page Left Intentionally

BLANK

A.1 OUTPUT ORGANIZATION

The results of an ABI run are organized in a compact, clearly identifiable and easy to follow manner. The following gives an overview of the organization of ABI output.

The output is subdivided into two primary parts:

1. Problem definition
2. Solution

Each separate entity of the *problem definition* or *solution* is placed in a *data block* with its own unique data block number and title. The data blocks and their identification numbers are shown in the following output overview. Data blocks 1 through 5 are used to reflect the input data, as interpreted by ABI.

The solution generally consists of several *stages*. Each stage refers to a *request for output* by the user. The printout of each stage is organized in the group of data blocks 100 through 111 (see the following output overview). For example data block 108 refers to the printout of stresses. If there are three stages to the solution, data block 108 for stresses appears three times, each time in its respective solution stage. The sequence of printout strictly follows the numerical order given in the output overview, regardless of the sequence of data entry and solution path.

Several features are used to enhance readability and compress the solution. If an item is not present, such as springs, the associated data block is automatically deleted. If loading is removed from a node, or at nodes where there are no loading, no printout is made. Printout is limited to items which exit for the stage at which printout is requested. The exception is for the solution items, where computed zero values of stresses, displacements and actions are printed.

A maximum of 80 columns are used for output organization. This allows printout on letter-size paper.

An optional pagination command, paginates the output to the user specified number of lines per page, with each page numbered and bearing the identification of the problem.

A.1.1 Output Overview

The computer output starts with the following blocks:

- A. user identification block
- B. software identification block
- C. current data identification

D. date and time of data execution

The preceding information is followed by data blocks listed below:

1-PROBLEM TITLE

2-UNITS

3-MATERIALS

- 3.1 CONCRETE PROPERTIES
- 3.2 MILD STEEL PROPERTIES
- 3.3 PRESTRESSING PROPERTIES

4-SECTION GEOMETRIES

5-FRAME GEOMETRY

- 5.1 NODES
- 5.2 FRAME ELEMENTS
 - 5.2.1 PRIMARY FRAME ELEMENTS
 - 5.2.2 SPRINGS
- 5.3 TENDONS
- 5.4 TRAVELERS

CONSTRUCTION AND SOLUTION

```

..... start of a stage
.....
100 - STAGE N      DAY D
101 - SOLUTION CONTROL DATA
102 - STRUCTURE
    102.1 ACTIVE NODES
    102.2 ACTIVE FRAME ELEMENT
    102.3 ACTIVE SPRINGS
    102.6 ACTIVE PRESTRESSING TENDONS
103 - TRAVELERS
    103.1 ACTIVE TRAVELERS AND THEIR POSITION
104 - LOADING
    104.1 EXTERNALLY APPLIED LOAD OF THIS INCREMENT
    104.2 ELEMENT TEMPERATURES APPLIED AT THIS INCREMENT
    104.3 TOTAL OF EXTERNALLY APPLIED LOADING AT THIS STAGE
    104.4 TOTAL OF ELEMENT TEMPERATURES AT THIS STAGE
105 - NODAL BOUNDARY CONDITIONS AND TOTAL DISPLACEMENTS
106 - TOTAL REACTIONS AT FIXED NODES
107 - FRAME ELEMENT ACTIONS
    107.1 PRIMARY ELEMENTS
    107.2 TRAVELER ACTIONS
108 - EXTREME FIBER STRESSES IN CONCRETE
109 - TENDON FORCES
    109.1 INDIVIDUAL TENDON ACTIONS
110 - TOTAL STATIC RESULTS FOR ELEMENTS
111 - CAMBER
..... start of next stage .....
```

Data blocks 100 through 111 will be repeated for each loading and construction stage.

```

-----
|               N A M E   O F   U S E R               |
|               ADDRESS OF USER                       |
|               ADDRESS OF USER                       |
|-----|
|               A D A P T   Structural Concrete Software System |
|               A D A P T - BRIDGE INCREMENTAL   ABI   (V-2.00   Jan 96) |
| 1733 Woodside Road,Suite 220, Redwood City, Calif 94061 Tel:415-306-2400 |
|-----|
Name of data file of this run : BRIDGE-1
Date of this run ..... :

```

1 - PROBLEM TITLE:

```

=====
ABI-GEN1 EXAMPLE THREE SPAN BRIDGE, BALANCED CANTILEVER CONSTRUCTION
END SPANS NEXT TO ABUTMENTS CONSTRUCTED ON FALSEWORK

```

2 - UNITS

```

=====
LB-IN

```

3 - MATERIALS

```

=====
3.1 CONCRETE PROPERTIES

```

```

-----
CONCRETE PROPERTIES: N = 1

```

```

CONCRETE NO.= 1
MODEL NO.   = 1

```

```

ULTIMATE STRENGTH  5.000E+03
CREEP COEFFICIENT  0.000E+00
SHRINKAGE STRAIN   0.000E+00

```

```

UNIT WEIGHT        8.970E-02
THERM EXPN COEFF   0.000E+00

```

3.2 MILD STEEL PROPERTIES

```

-----
N = 1

```

MILD STEEL NO.	ELASTIC MODULUS	PERCENTAGE IN CROSS SECTION	THERMAL EXPANSION COEFFICIENT
1	2.90000E+07	2.00000E-02	0.00000E+00

3.3 PRESTRESSING TENDON MATERIAL PROPERTIES

```

-----
N= 1

```

STEEL NO.	ELASTIC MODULUS	CURVATURE FRICTION	WOBBLE FRICTION	ULTIMATE STRESS	RELAX COEFF	THERMAL COEFF
1	2.800E+07	7.000E-02	1.000E+-4	2.700E+05	5.000E+02	0.000E+00

4 - SECTION GEOMETRIES

```

=====
N = 8

```

INPUT OR COMPUTED SECTION PROPERTIES

SECTION NO.	1	2	3	4	5
CROSS SECTION AREA	3.456E+03	3.564E+03	3.996E+03	4.860E+03	6.156E+03
MOMENT OF INERTIA	1.493E+06	1.637E+06	2.308E+06	4.152E+06	8.438E+06
DISTANCE CG TO TOP	3.600E+01	3.713E+01	4.163E+01	5.063E+01	6.413E+01
DISTANCE CG TO BOT	3.600E+01	3.713E+01	4.163E+01	5.063E+01	6.413E+01
SHRIN FACTOR, TOP	1.000E+00	1.000E+00	1.000E+00	1.000E+00	1.000E+00
SHRIN FACTOR, BOT	1.000E+00	1.000E+00	1.000E+00	1.000E+00	1.000E+00

5 - FRAME GEOMETRY

=====

5.1 NODES

N = 38

NODE	X-COORD	Y-COORD	NODE	X-COORD	Y-COORD
1	.000	960.000	20	3096.000	960.000
2	300.000	960.000	21	3360.000	957.720
3	300.000	360.000	22	3624.000	951.000
4	600.000	960.000	23	3888.000	939.720

5.2 FRAME ELEMENTS

5.2.1 PRIMARY FRAME ELEMENTS.....

N= 31

ELEMENT NO.	NODE I	NODE J	CONCR TYPE	SECT TYPE	STEEL TYPE	CASTING DAY
1	1	2	1	1	1	1
2	2	4	1	1	1	12
3	4	6	1	1	1	12
4	6	7	1	2	1	124
5	7	8	1	3	1	124

5.2.2 SPRINGS.....

N= 4

SPRING NO.	NODE I	NODE J	KE	KR
34	2	3	1.0000E+05	1.0000E+04
35	4	5	1.0000E+05	1.0000E+04

5.3 TENDONS

N= 11

TENDON NO.= 11 MATERIAL= 1 AREA= 12.40000

T.P. NO.	GLOBAL X-	COORDINATES Y-	ANGLE (RADIAN)	NODE NO.	OFFSET FROM X-	NODE Y-
1	4.200E+03	9.840E+02	0.000E+00	25	0.000E+00	6.000E+01
2	4.248E+03	9.836E+02	3.590E-02	26	0.000E+00	5.964E+01
3	4.512E+03	9.722E+02	4.067E-03	30	0.000E+00	3.248E+01
4	4.776E+03	9.618E+02	1.628E-02	31	0.000E+00	1.084E+01
5	5.040E+03	9.558E+02	1.625E-02	32	0.000E+00	-1.942E+00
6	5.304E+03	9.540E+02	8.375E-03	33	0.000E+00	-5.993E+00
7	5.400E+03	9.542E+02	4.583E-03	34	0.000E+00	-5.833E+00
8	5.700E+03	9.560E+02	6.944E-03	36	0.000E+00	-3.958E+00
9	6.000E+03	9.600E+02	0.000E+00	38	0.000E+00	0.000E+00

SEGMENT NO.	T.P. I	T.P. J	NODE I	NODE J	X-LENGTH	Y-LENGTH	TOTAL LENGTH	ANGLE (RADIAN)
1	1	2	25	26	4.800E+01	-3.556E-01	4.800E+01	1.795E-02
2	2	3	26	30	2.640E+02	-1.144E+01	2.642E+02	1.998E-02
3	3	4	30	31	2.640E+02	-1.036E+01	2.642E+02	1.017E-02
4	4	5	31	32	2.640E+02	-6.062E+00	2.641E+02	1.627E-02
5	5	6	32	33	2.640E+02	-1.771E+00	2.640E+02	1.231E-02
6	6	7	33	34	9.600E+01	1.600E-01	9.600E+01	6.479E-03
7	7	8	34	36	3.000E+02	1.875E+00	3.000E+02	5.764E-03
8	8	9	36	38	3.000E+02	3.958E+00	3.000E+02	3.472E-03

TOTALS							1.801E+03	9.240E-02
--------	--	--	--	--	--	--	-----------	-----------

5.4 TRAVELERS

 NUMBER OF TRAVELERS N = 2

TRAVELER NO.	NODE COUNT	SECTION AREA	MOMENT OF INERTIA	ELASTIC MODULUS	TOTAL WEIGHT
1	3	1.000E+02	2.200E+04	3.000E+07	1.000E+04
2	3	1.000E+02	2.200E+04	3.000E+07	1.000E+04

=====
 == CONSTRUCTION AND SOLUTION AT DAY 3000 STAGE 7 ==
 =====

100 - SOLUTION CONTROL DATA

=====

CURRENT TEMPERATURE	=	75.0
X-DIRECTION GRAVITY MULTIPLIER	=	0.0
Y-DIRECTION GRAVITY MULTIPLIER	=	-1.0
STRESS CONVERGENCE FACTOR	=	1.0

102 - STRUCTURE

102.1 ACTIVE NODES

1	2	4	6	7	8	9	10	11	12
13	14	15	16	17	18	19	20	21	22
23	24	25	26	27	28	29	30	31	32
33	34	36	38						

102.2 ACTIVE FRAME ELEMENTS

1	2	3	4	5	6	7	8	9	10
11	12	13	14	15	16	17	18	19	20
21	22	23	24	25	26	27	29	30	32
33									

102.3 ACTIVE SPRINGS

2	3	22	45
---	---	----	----

102.6 ACTIVE PRESTRESSING TENDONS

1	3	8	12	14	16
---	---	---	----	----	----

103 - TRAVELERS

103.1 ACTIVE TRAVELERS AND THEIR POSITIONS

TRAVELER 1	AT NODES	3	4	5
TRAVELER 2	AT NODES	12	22	23 28

104 - LOADING

=====

104.1 EXTERNALLY APPLIED LOAD OF THIS I N C R E M E N T

NODE	X-FORC	Y-FORC	Z-MOMT	X-DISP	Y-DISP	Z-ROTN
2	0.0000E+00	1.0000E+02	-4.0000E+01	0.0000E+00	0.0000E+00	0.0000E+00
12	0.0000E+00	2.0000E+02	-4.0000E+01	0.0000E+00	0.0000E+00	0.0000E+00
22	0.0000E+00	3.0000E+02	-2.0000E+01	0.0000E+00	0.0000E+00	0.0000E+00

104.2 ELEMENT TEMPERATURES APPLIED AT THIS I N C R E M E N T

FRAME ELEMENT	TOP FIBER TEMPERATURE	BOTTOM FIBER TEMPERATURE
5	90.0	60.0
17	80.0	40.0

104.3 T O T A L OF EXTERNALLY APPLIED LOADING AT THIS S T A G E

NODE	X-FORC	Y-FORC	Z-MOMT	X-DISP	Y-DISP	Z-ROTN
4	0.0000E+00	-1.0000E+01	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
26	1.0000E+02	-1.0000E+02	4.0000E+01	0.0000E+00	0.0000E+00	0.0000E+00
36	0.0000E+00	-1.0000E+01	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00

104.4 TOTAL OF ELEMENT TEMPERATURES AT THIS S T A G E

FRAME ELEMENT	TOP FIBER TEMPERATURE	BOTTOM FIBER TEMPERATURE
5	90.0	60.0
17	80.0	40.0

105 - NODAL BOUNDARY CONDITIONS AND TOTAL DISPLACEMENTS

=====

Legend: 0 = Free ; 1 = Fixed

NODE	U(X)	U(Y)	R(Z)	X-DISP	Y-DISP	Z-ROTN
1	0	1	0	7.2826E-01	0.0000E+00	-1.1639E-03
2	0	0	0	6.7488E-01	-3.1222E-01	-7.9435E-04
4	0	0	0	6.2149E-01	-4.7075E-01	-3.6615E-04
6	0	0	0	7.5552E-01	6.6293E+00	-1.1832E-02
7	0	0	0	6.4461E-01	3.7952E+00	-9.7553E-03
8	0	0	0	4.7457E-01	1.6335E+00	-6.5984E-03
9	0	0	0	2.7957E-01	3.4222E-01	-3.1508E-03
10	0	0	0	1.1821E-01	-7.2421E-02	7.8462E-05

106 - TOTAL REACTIONS AT FIXED NODES

=====

NODE	X-FORCE	Y-FORCE	Z-MOMENT
1	0.0000E+00	-1.2432E+05	0.0000E+00
15	8.4631E+04	-1.4340E+06	-3.1388E+07
29	-8.4531E+04	-1.4121E+06	3.1713E+07
38	0.0000E+00	-1.2526E+05	0.0000E+00
TOTAL	1.0000E+02	-3.0957E+06	3.2451E+05

107 - FRAME ELEMENT ACTIONS

107.1 PRIMARY ELEMENTS

ELEMENT NO	BENDING (END I)	BENDING (END J)	SHEAR FORCE	AXIAL FORCE
1	2.9233E-09	1.2518E+07	4.1726E+04	-2.7358E+06
2	1.2517E+07	2.8342E+06	-3.2276E+04	-2.7359E+06
3	2.8339E+06	-4.9542E+06	-8.1127E+04	-2.7360E+06
4	5.3756E+07	3.6725E+07	-6.4511E+04	-4.4521E+06
5	1.0723E+08	1.1671E+08	3.5891E+04	-7.2495E+06
6	2.0403E+08	2.5304E+08	1.8550E+05	-1.0650E+07
7	3.6279E+08	4.6650E+08	3.9214E+05	-1.3126E+07
8	4.7163E+08	4.4404E+08	-5.7487E+05	-1.3212E+07

107.2 SPRING ACTIONS

SPRING	AXIAL FORCE	BENDING MOMENT
34	-2.1077E+04	-1.5181E+01
35	-3.4659E+04	-1.4110E+01
36	-3.4417E+04	0.0000E+00
37	-2.1191E+04	0.0000E+00

108 - EXTREME FIBER STRESSES IN CONCRETE

ELEMENT	I-TOP	I-BOT	J-TOP	J-BOT	AXIAL
1	-7.9057E+02	-7.9057E+02	-1.0919E+03	-4.8924E+02	-7.9057E+02
2	-1.0919E+03	-4.8931E+02	-8.5888E+02	-7.2236E+02	-7.9062E+02
3	-8.5871E+02	-7.2213E+02	-6.7119E+02	-9.0965E+02	-7.9042E+02
4	-2.4642E+03	-3.0424E+01	-2.0786E+03	-4.1604E+02	-1.2473E+03
5	-3.7430E+03	1.1967E+02	-3.9137E+03	2.9037E+02	-1.8117E+03

109 - TENDON FORCES

109.1 INDIVIDUAL TENDON ACTIONS

TENDON 1 TOTAL RESULTS

SEG NO	TOTAL FORCE	PERCENT LOSS	FRAME EL NO	BENDING (END I)	BENDING (END J)	SHEAR FORCE	AXIAL FORCE
1	2.449E+06	10.49	7	-1.084E+08	-1.469E+08	-1.456E+05	2.445E+06
2	2.489E+06	9.01	8	-1.494E+08	-1.494E+08	0.000E+00	2.489E+06
3	2.468E+06	9.81	9	-1.481E+08	-1.481E+08	0.000E+00	2.468E+06
4	2.427E+06	11.31	10	-1.456E+08	-1.075E+08	1.442E+05	2.422E+06

TENDON 2 TOTAL RESULTS

SEG NO	TOTAL FORCE	PERCENT LOSS	FRAME EL NO	BENDING (END I)	BENDING (END J)	SHEAR FORCE	AXIAL FORCE
1	2.593E+06	5.24	6	-8.556E+07	-1.148E+08	-1.107E+05	2.590E+06
2	2.612E+06	4.54	7	-1.156E+08	-1.567E+08	-1.552E+05	2.607E+06
3	2.633E+06	3.76	8	-1.580E+08	-1.580E+08	0.000E+00	2.633E+06
4	2.611E+06	4.56	9	-1.567E+08	-1.567E+08	0.000E+00	2.611E+06
5	2.589E+06	5.36	10	-1.554E+08	-1.147E+08	1.539E+05	2.585E+06
6	2.573E+06	5.97	11	-1.139E+08	-8.490E+07	1.098E+05	2.570E+06

109.2 COMBINED ACTION AND FORCE OF ALL TENDONS

ELEMENT NO	NO OF TENDONS	BENDING (END I)	BENDING (END J)	SHEAR FORCE	AXIAL FORCE
1	1	-1.8050E-09	1.5751E+07	5.2505E+04	0.0000E+00
2	1	1.5752E+07	9.9260E+06	-1.9418E+04	5.5180E-02
3	1	9.9260E+06	5.4957E+06	-4.6149E+04	1.0629E-01
4	3	5.4957E+06	-2.2417E+07	-1.0573E+05	-9.1298E+02
5	4	-2.2417E+07	-7.5285E+07	-2.0019E+05	-5.0957E+03
6	4	-7.5285E+07	-1.8119E+08	-4.0077E+05	-1.2352E+05
7	5	-1.8119E+08	-3.2110E+08	-5.2903E+05	-1.3799E+05

110 TOTAL STATIC RESULTS FOR ELEMENTS

ELEMENT NO	NO OF TENDONS	BENDING (END I)	BENDING (END J)	SHEAR FORCE	AXIAL FORCE
1	1	-1.8050E-09	1.5751E+07	5.2505E+04	0.0000E+00
2	1	1.5752E+07	9.9260E+06	-1.9418E+04	5.5180E-02
3	1	9.9260E+06	5.4957E+06	-4.6149E+04	1.0629E-01
4	3	5.4957E+06	-2.2417E+07	-1.0573E+05	-9.1298E+02
5	4	-2.2417E+07	-7.5285E+07	-2.0019E+05	-5.0957E+03

111 - CAMBER

NODE	X-DIRECTION	Y-DIRECTION	ROTATION
1	-4.5806E-01	0.0000E+00	-1.3253E-04
2	-4.0468E-01	-5.0331E-02	-2.3825E-04
3	0.0000E+00	0.0000E+00	0.0000E+00
4	-3.5129E-01	-1.4881E-01	-4.0261E-04
5	0.0000E+00	0.0000E+00	0.0000E+00
6	-7.5552E-01	-6.6293E+00	1.1832E-02
7	-6.4461E-01	-3.7952E+00	9.7553E-03

A.2 EXAMPLE

The following is an example of input and output of the simple cantilever problem described next:

The sample input and output is for a simple non-prestressed cantilever with uniform rectangular cross-section subject to a concentrated loading at its tip. For the purposes of illustration, the weight of the cantilever is specified as zero. Hence the actions (moments and shears) and the deformations will be due to the concentrated load only.

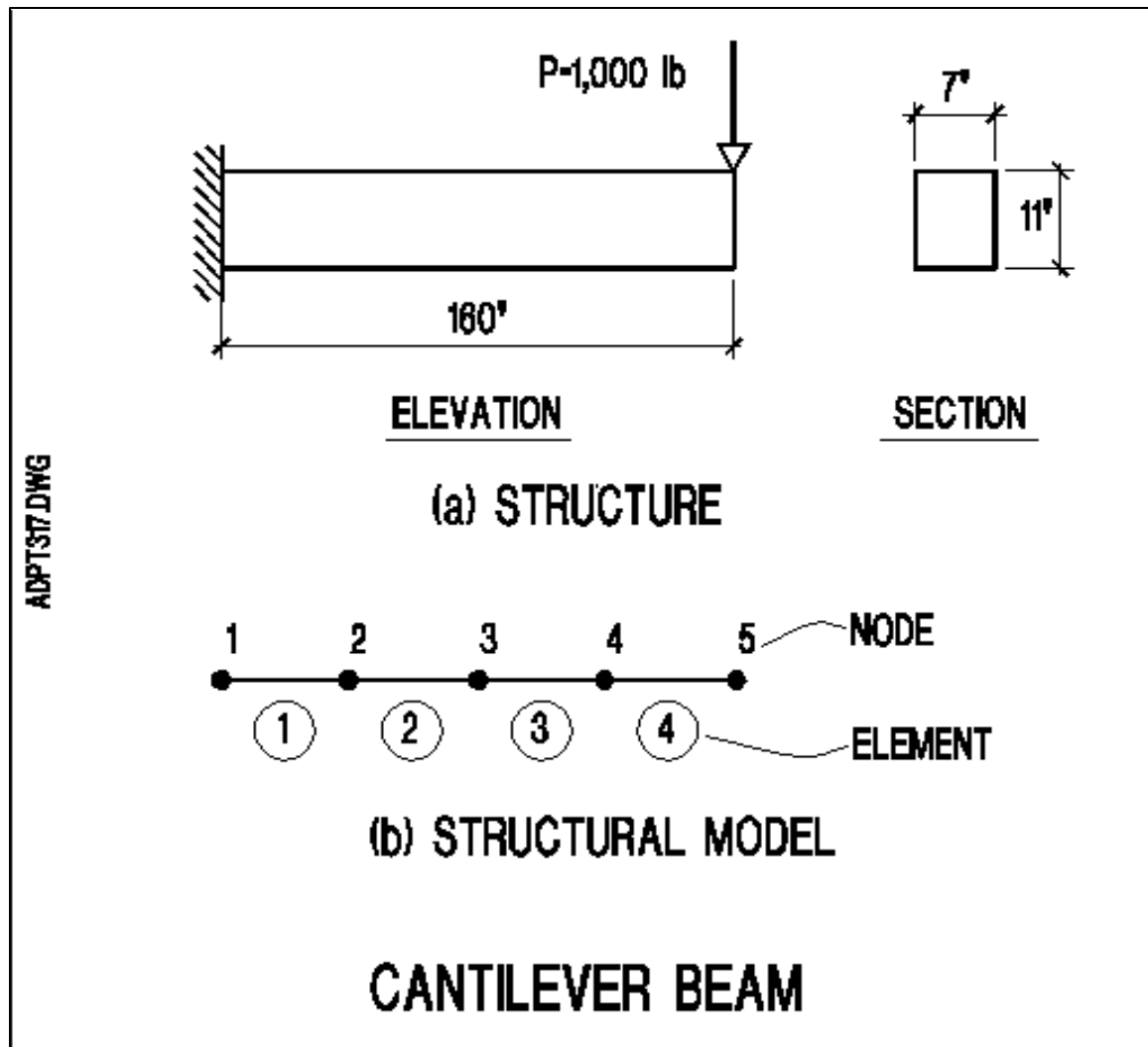


FIGURE A.2-1

The cantilever is cast at day zero; loaded at day 100, at which time its moments and deformations are calculated. Without changing the loading, two other solutions are

obtained, one at 1000 days and the other at 10,000 days. Since the loading is not changed, the actions will be the same for the different ages, but the deformations will increase. The change in deformation is triggered by the non-zero creep (C) and shrinkage (S) coefficients entered under "CONCRETE PARAMETERS" command.

A.3 INPUT FILE

```

=====
;      ADAPT-BRIDGE-INCREMENTAL      ABI      SOFTWARE MANUAL EXAMPLE
=====
; Name of this file : CANT.INP

START
TITLE N=2
      CANT.INP      CANTILEVER BEAM EXAMPLE FIXED AT LEFT
                    FOUR ELEMENTS MODELING units: lb-inch

UNITS U=USA
CONCRETE PARAMETERS N=1 ! 1 M=ACI
MESH INPUT
NODES N=5
      1 X=0 Y=0 ! 5 X=160 Y=0 G=1,5

      CONCRETE PROPERTIES N=1
        1 Fpc=5000 Cr=3.5 Sh=0.0005 W=0.0
      MILD STEEL PROPERTIES N=1
        1 Es=29000000 P=0.05
      SECTION PROPERTIES N=1
        1 D=11 B=7
      ELEMENTS N=4
      FRAME N=4
        1,1,2 C=1 X=1 St=1 Day=0 G=1,4,1,1,1

MESH COMPLETE

SET Day=100

CHANGE STRUCTURE
  RESTRAINTS
    1 R=1,1,1

  BUILD N=1,4,1
CHANGE COMPLETE

LOADING
  N=5 F=0, -1000,0

SOLVE      ! OUTPUT ;..... stage 1.....
SOLVE Day=1000 Steps=10 ! OUTPUT ;..... stage 2.....
SOLVE Day=10000 Steps=10 ! OUTPUT ;..... stage 3.....
STOP

```

A.4 OUTPUT LISTING

```

-----
|          A D A P T   Structural Concrete Software System          |
|          A D A P T - BRIDGE INCREMENTAL   ABI   (V-2.00   Jan 96) |
| 1733 Woodside Road,Suite 220, Redwood City, Calif 94061 Tel:415-306-2400 |
|-----|
Name of data file of this run : cant
Date and time of this run ... :

1 - PROBLEM TITLE:
=====
      EXAMPLE:  CANTILEVER BEAM, FIXED AT LEFT END
                FOUR ELEMENT MODELING      UNITS: LB-INCH

2 - UNITS
=====
LB-IN

3 - MATERIALS
=====
3.1 CONCRETE PROPERTIES
-----
CONCRETE PROPERTIES: N =   1

CONCRETE NO.           1
MODEL NO.              1

ULTIMATE STRENGTH      5.000E+03
CREEP COEFFICIENT      3.500E+00
SHRINKAGE STRAIN       5.000E-04

UNIT WEIGHT            0.000E+00
THERM EXPN COEFF       0.000E+00

3.2 MILD STEEL PROPERTIES
-----
N =   1

      MILD STEEL      ELASTIC      PERCENTAGE IN      THERMAL EXPANSION
      NO.             MODULUS      CROSS SECTION      COEFFICIENT

           1      2.90000E+07      5.00000E-02      0.00000E+00

4 - SECTION GEOMETRIES
=====
N =   1

INPUT OR COMPUTED SECTION PROPERTIES

SECTION NO.           1
CROSS SECTION AREA    7.700E+01
MOMENT OF INERTIA     7.764E+02
DISTANCE CG TO TOP    5.500E+00
DISTANCE CG TO BOT    5.500E+00
SHRIN FACTOR, TOP     1.000E+00
SHRIN FACTOR, BOT     1.000E+00

```

5 - FRAME GEOMETRY

=====

5.1 NODES

N = 5

NODE	X-COORD	Y-COORD	NODE	X-COORD	Y-COORD
1	.000	.000	4	120.000	.000
2	40.000	.000	5	160.000	.000
3	80.000	.000			

5.2 FRAME ELEMENTS

5.2.1 PRIMARY FRAME ELEMENTS

N = 4

ELEMENT NO.	NODE I	NODE J	CONCR TYPE	SECT TYPE	STEEL TYPE	CASTING DAY
1	1	2	1	1	1	0
2	2	3	1	1	1	0
3	3	4	1	1	1	0
4	4	5	1	1	1	0

```
=====
=== CONSTRUCTION AND SOLUTION AT DAY 100 STAGE 1 ===
=====
```

100 - SOLUTION CONTROL DATA

=====

```
CURRENT TEMPERATURE..... = 70.00
X-DIRECTION GRAVITY MULTIPLIER = .00
Y-DIRECTION GRAVITY MULTIPLIER = -1.00
STRESS CONVERGENCE FACTOR..... = .70
```

102 - STRUCTURE

=====

102.1 ACTIVE NODES

```
1  2  3  4  5
```

102.2 ACTIVE FRAME ELEMENTS

```
1  2  3  4
```

104 - LOADING

=====

104.1 EXTERNALLY APPLIED LOAD OF THIS I N C R E M E N T

NODE	X-FORC	Y-FORC	Z-MOMT	X-DISP	Y-DISP	Z-ROTN
5	0.0000E+00	-2.0000E+02	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00

104.3 T O T A L OF EXTERNALLY APPLIED LOADING AT THIS S T A G E

NODE	X-FORC	Y-FORC	Z-MOMT	X-DISP	Y-DISP	Z-ROTN
5	0.0000E+00	-2.0000E+02	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00

105 - NODAL BOUNDARY CONDITIONS AND TOTAL DISPLACEMENTS

=====

Legend: 0 = Free ; 1 = Fixed

NODE	U(X)	U(Y)	R(Z)	X-DISP	Y-DISP	Z-ROTN
1	1	1	1	0.0000E+00	0.0000E+00	0.0000E+00
2	0	0	0	0.0000E+00	-6.6324E-03	-3.1655E-04
3	0	0	0	0.0000E+00	-2.4118E-02	-5.4265E-04
4	0	0	0	0.0000E+00	-4.8838E-02	-6.7831E-04
5	0	0	0	0.0000E+00	-7.7177E-02	-7.2353E-04

106 - TOTAL REACTIONS AT FIXED NODES

=====

NODE	X-FORCE	Y-FORCE	Z-MOMENT
1	0.0000E+00	-2.0000E+02	-3.2000E+04
TOTAL	0.0000E+00	-2.0000E+02	-3.2000E+04

107 - FRAME ELEMENT ACTIONS

=====

107.1 PRIMARY ELEMENTS

ELEMENT NO	BENDING (END I)	BENDING (END J)	SHEAR FORCE	AXIAL FORCE
1	-3.2000E+04	-2.4000E+04	2.0000E+02	0.0000E+00
2	-2.4000E+04	-1.6000E+04	2.0000E+02	0.0000E+00
3	-1.6000E+04	-8.0000E+03	2.0000E+02	0.0000E+00
4	-8.0000E+03	-6.6001E-11	2.0000E+02	0.0000E+00

108 - STRESSES

=====

ELEMENT	I-TOP	I-BOT	J-TOP	J-BOT	AXIAL
1	2.2596E+02	-2.2596E+02	1.6947E+02	-1.6947E+02	0.0000E+00
2	1.6947E+02	-1.6947E+02	1.1298E+02	-1.1298E+02	0.0000E+00
3	1.1298E+02	-1.1298E+02	5.6490E+01	-5.6490E+01	0.0000E+00
4	5.6490E+01	-5.6490E+01	4.6605E-13	-4.6605E-13	0.0000E+00

```

=====
=== CONSTRUCTION AND SOLUTION AT DAY 1000 STAGE 2 ===
=====

```

100 - SOLUTION CONTROL DATA

=====

```

CURRENT TEMPERATURE..... = 70.00
X-DIRECTION GRAVITY MULTIPLIER = .00
Y-DIRECTION GRAVITY MULTIPLIER = -1.00
STRESS CONVERGENCE FACTOR..... = .70

```

102 - STRUCTURE

=====

102.1 ACTIVE NODES

1	2	3	4	5
---	---	---	---	---

102.2 ACTIVE FRAME ELEMENTS

1	2	3	4
---	---	---	---

104 - LOADING

104.1 EXTERNALLY APPLIED LOAD OF THIS I N C R E M E N T

NODE	X-FORC	Y-FORC	Z-MOMT	X-DISP	Y-DISP	Z-ROTN
5	0.0000E+00	-2.0000E+02	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00

104.3 T O T A L OF EXTERNALLY APPLIED LOADING AT THIS S T A G E

NODE	X-FORC	Y-FORC	Z-MOMT	X-DISP	Y-DISP	Z-ROTN
5	0.0000E+00	-2.0000E+02	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00

105 - NODAL BOUNDARY CONDITIONS AND TOTAL DISPLACEMENTS

Legend: 0 = Free ; 1 = Fixed

NODE	U(X)	U(Y)	R(Z)	X-DISP	Y-DISP	Z-ROTN
1	1	1	1	0.0000E+00	0.0000E+00	0.0000E+00
2	0	0	0	-6.0200E-03	-2.3624E-02	-1.1275E-03
3	0	0	0	-1.2040E-02	-8.5905E-02	-1.9329E-03
4	0	0	0	-1.8060E-02	-1.7396E-01	-2.4161E-03
5	0	0	0	-2.4080E-02	-2.7489E-01	-2.5771E-03

106 - TOTAL REACTIONS AT FIXED NODES

NODE	X-FORCE	Y-FORCE	Z-MOMENT
1	1.2340E+01	-1.9941E+02	-3.1905E+04
TOTAL	1.2340E+01	-1.9941E+02	-3.1905E+04

107 - FRAME ELEMENT ACTIONS

107.1 PRIMARY ELEMENTS

ELEMENT NO	BENDING (END I)	BENDING (END J)	SHEAR FORCE	AXIAL FORCE
1	-3.1905E+04	-2.3929E+04	1.9941E+02	1.2340E+01
2	-2.3929E+04	-1.5952E+04	1.9941E+02	1.2340E+01
3	-1.5952E+04	-7.9762E+03	1.9941E+02	1.2340E+01
4	-7.9762E+03	-1.9470E-10	1.9941E+02	1.2340E+01

108 - STRESSES

ELEMENT	I-TOP	I-BOT	J-TOP	J-BOT	AXIAL
1	2.2578E+02	-2.2110E+02	1.6992E+02	-1.6524E+02	2.3425E+00
2	1.6992E+02	-1.6524E+02	1.1406E+02	-1.0938E+02	2.3425E+00
3	1.1406E+02	-1.0938E+02	5.8202E+01	-5.3517E+01	2.3425E+00
4	5.8202E+01	-5.3517E+01	2.3425E+00	2.3425E+00	2.3425E+00

```

=====
===  CONSTRUCTION AND SOLUTION    AT DAY10000    STAGE  3    ===
=====

100 - SOLUTION CONTROL DATA
=====
CURRENT TEMPERATURE..... = 70.00
X-DIRECTION GRAVITY MULTIPLIER = .00
Y-DIRECTION GRAVITY MULTIPLIER = -1.00
STRESS CONVERGENCE FACTOR..... = .70

102 - STRUCTURE
=====
102.1 ACTIVE NODES
-----
1    2    3    4    5

102.2 ACTIVE FRAME ELEMENTS
-----
1    2    3    4

104 - LOADING
=====
104.1 EXTERNALLY APPLIED LOAD OF THIS I N C R E M E N T
-----

NODE    X-FORC    Y-FORC    Z-MOMT    X-DISP    Y-DISP    Z-ROTN
5  0.0000E+00 -2.0000E+02  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00

104.3  T O T A L  OF EXTERNALLY APPLIED LOADING AT THIS S T A G E
-----

NODE    X-FORC    Y-FORC    Z-MOMT    X-DISP    Y-DISP    Z-ROTN
5  0.0000E+00 -2.0000E+02  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00

105 - NODAL BOUNDARY CONDITIONS AND TOTAL DISPLACEMENTS
=====
Legend: 0 = Free ; 1 = Fixed

      NODE      U(X)      U(Y)      R(Z)      X-DISP      Y-DISP      Z-ROTN
      1         1         1         1      0.0000E+00  0.0000E+00  0.0000E+00
      2         0         0         0     -6.8683E-03 -2.5656E-02 -1.2245E-03
      3         0         0         0     -1.3737E-02 -9.3295E-02 -2.0991E-03
      4         0         0         0     -2.0605E-02 -1.8892E-01 -2.6239E-03
      5         0         0         0     -2.7473E-02 -2.9854E-01 -2.7989E-03

106 - TOTAL REACTIONS AT FIXED NODES
=====
      NODE    X-FORCE    Y-FORCE    Z-MOMENT
      1  5.7569E-12 -2.0000E+02 -3.2000E+04
TOTAL  5.7569E-12 -2.0000E+02 -3.2000E+04

```

107 - FRAME ELEMENT ACTIONS

=====

107.1 PRIMARY ELEMENTS

ELEMENT NO	BENDING (END I)	BENDING (END J)	SHEAR FORCE	AXIAL FORCE
1	-3.2000E+04	-2.4000E+04	2.0000E+02	5.7696E-12
2	-2.4000E+04	-1.6000E+04	2.0000E+02	5.7696E-12
3	-1.6000E+04	-8.0000E+03	2.0000E+02	-4.1496E-12
4	-8.0000E+03	9.3709E-13	2.0000E+02	5.8549E-12

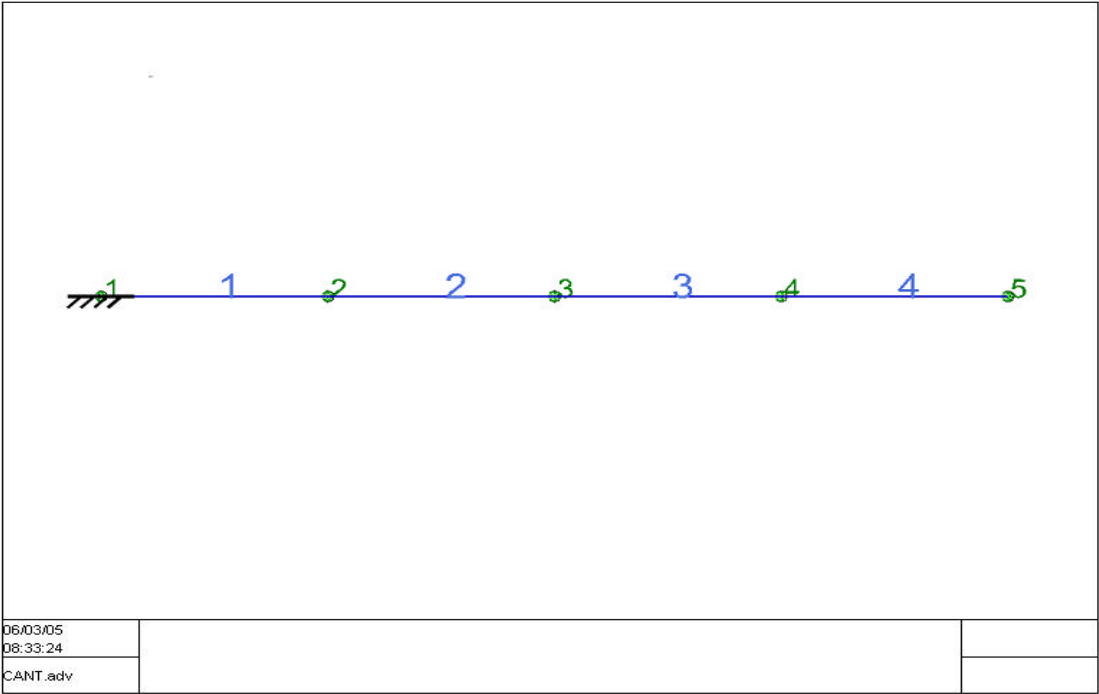
108 - STRESSES

=====

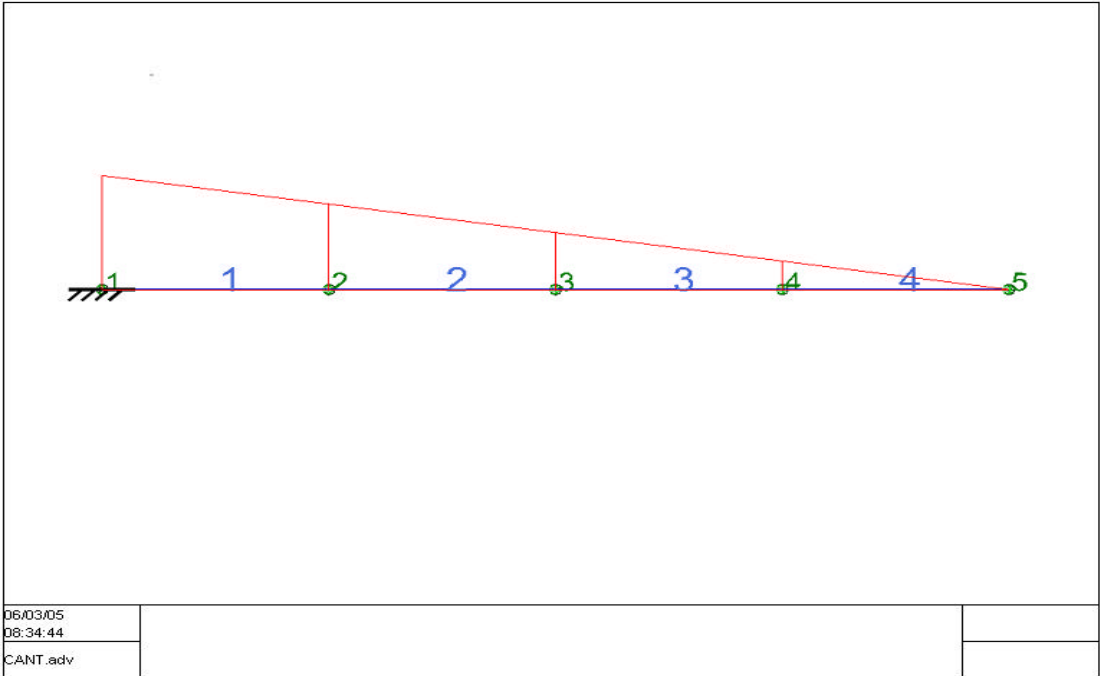
ELEMENT	I-TOP	I-BOT	J-TOP	J-BOT	AXIAL
1	2.2638E+02	-2.2140E+02	1.7041E+02	-1.6543E+02	2.4898E+00
2	1.7041E+02	-1.6543E+02	1.1444E+02	-1.0946E+02	2.4898E+00
3	1.1444E+02	-1.0946E+02	5.8463E+01	-5.3483E+01	2.4898E+00
4	5.8463E+01	-5.3483E+01	2.4898E+00	2.4898E+00	2.4898E+00

A.5 GRAPHICAL DISPLAY OF OUTPUT

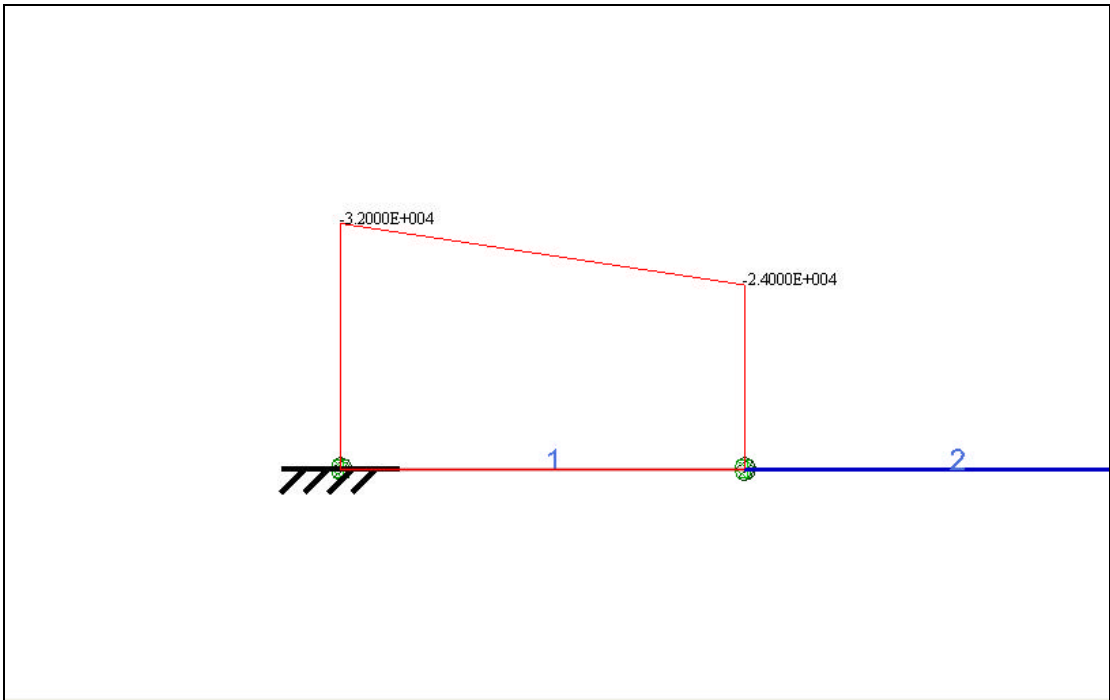
The following illustrates several graphical displays of the output generated by ABI's print option.



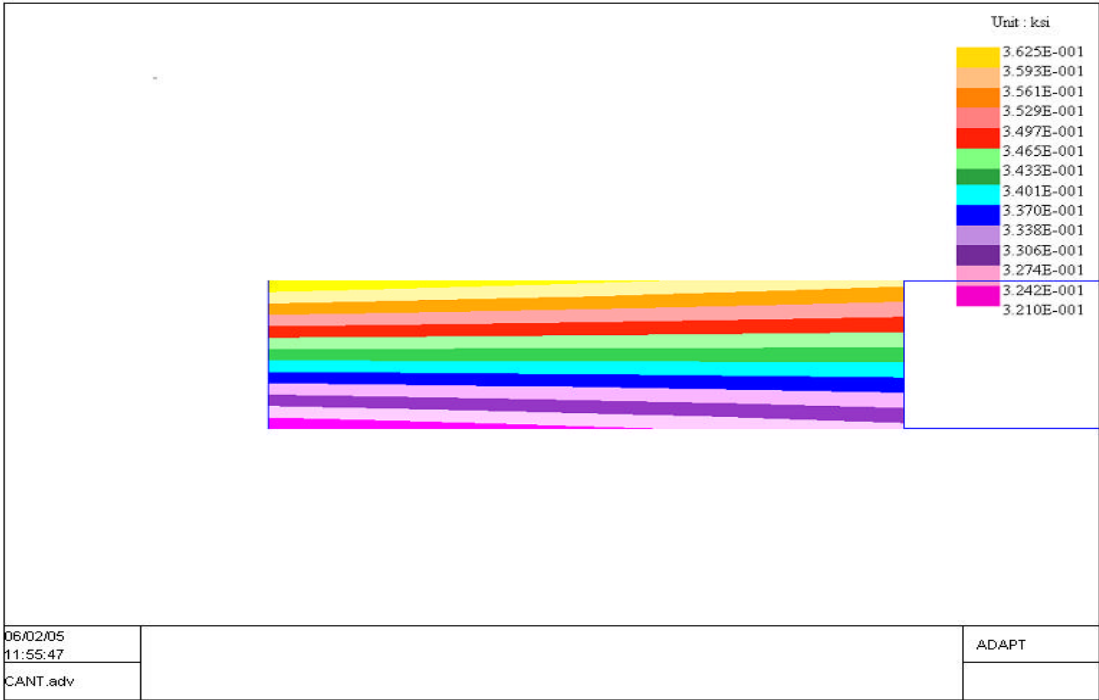
DISCRETIZATION OF CANTILEVER EXAMPLE
FIGURE A.5-1



MOMENTS IN CANTILEVER EXAMPLE
FIGURE A.5-2



MOMENT IN ELEMENT NEXT TO SUPPORT
FIGURE A.5-3



STRESSES IN ELEMENT NEXT TO SUPPORT
FIGURE A.5-4

A.6 INPUT/OUTPUT GRAPHICAL DISPLAY

The graphical display for both the input and output of the program is described in a separate package entitled “ABI-Viewer Interface”.

B. MODELING OF CONCRETE'S TIME DEPENDENT BEHAVIOR

B.1 STRAIN COMPONENTS AND THE SUPERPOSITION METHOD.....	4
B.1.1 Mechanical Strain	5
B.1.2 Aging Strain	6
B.1.3 Creep Strain	6
B.1.4 Shrinkage Strain	6
B.1.5 Temperature Strain	8
B.2 PREDICTION OF TIME DEPENDENT MATERIAL PROPERTIES	8
B.2.1 ACI Committee 209 Recommendations.....	8
A. Strength and stiffness	8
B. Creep strain	9
C. Shrinkage strain	10
B.2.2 CEB - FIP Committee Recommendations	11
A. Strength, modulus of elasticity and aging of concrete.....	11
B. Creep strain	13
C. Shrinkage strain	15
B.3 MATHEMATICAL MODELING OF THE CREEP STRAIN COMPONENT.....	15
B.3.1 Theoretical Background.....	15
B.3.2 Calculation of the Creep Strain Increment.....	18
A. Constant stress and constant material parameters	18
B. Linear stress and constant material parameters	19
C. Linear stress and linear material parameters	20
B.3.3 Determination of creep compliance coefficients	21

This Page Left Intentionally

BLANK

Concrete is unique among structural materials in that it undergoes complex physical and chemical changes over time, resulting in deformations and constitutive properties which are time dependent under practical service conditions. These time dependent phenomena in concrete are some of the most significant factors influencing the structural behavior of segmentally erected prestressed concrete bridges. Accurate consideration of time dependent concrete behavior is necessary for the accurate prediction of stresses and deflections in the structure at all load levels. Nonlinear concrete behavior such as cracking, crushing, and the nonlinear stress-strain relationship of the concrete also influence structural behavior, but are excluded from this investigation of purely time dependent behavior.

Time dependent behavior of concrete may be of three different classifications: time dependent material properties resulting in time dependent stress-strain relationships and producing fictitious aging strains; time dependent stress originated (creep) strain; and time dependent non-stress originated (shrinkage and temperature change) strains. Strictly speaking, these three classes of effects are not independent. However, in practice, they may be treated as independent of each other and hence additive.

Material properties influenced by time include the strength $f_c(t)$ and the stiffness $E_c(t)$. The strength and stiffness increase significantly during the first month after casting, then they increase more slowly over the remainder of life of the structure. Aging strains are fictitious strains used in an analysis to allow for time dependent stiffness.

Creep and shrinkage strains in concrete are influenced by a number of factors depending on the mix design, the loading history, and the environment. Predicted displacements and internal stresses in the structure may be significantly in error if these strains are neglected. Of these strain components, creep strain is by far the most difficult to predict because it is stress originated and depends on the entire stress history of the concrete as well as other factors.

The analysis of a structure for the time dependent effects of creep of concrete requires the solution of two important problems which are addressed in Sections B.2 and B.3 of this chapter:

1. Given the concrete mix design, the proportions of the structural elements and the environmental conditions under which the structure is built, find a relationship expressing the creep strain at any time under constant stress applied at any loading age. Since the mechanisms of creep and shrinkage in concrete are not understood completely, and correlations between field and laboratory studies are imprecise even under carefully controlled conditions, no theory has explained adequately all the observed information. Therefore two alternative models for predicting time dependent creep and shrinkage strains are reviewed in Section B.2 of this chapter.

2. Given this relationship for time dependent strains under constant stress, find the actual stress and strain history in the structure under varying loads. Solution of this problem requires numerical integration over time of the differential equation of which the above relationship is the kernel. The choice of method to use for this numerical integration is influenced by considerations of accuracy, convergence and solution economy. A refined numerical integration scheme for use in a step-by-step finite element analysis is derived in Section B.3 of this chapter.

B.1 STRAIN COMPONENTS AND THE SUPERPOSITION METHOD

An important assumption in studying and predicting the time dependent behavior of concrete structures is that the total strain in the concrete may be considered as a superposition of several independent components caused by different phenomena. Even though creep and shrinkage are not strictly independent, and even under simple isothermal conditions the presence of shrinkage increases the magnitude of creep, this commonly accepted assumption has been experimentally verified. It has been used by many investigators to study the time dependent behavior of concrete structures.

In this investigation, the total uniaxial concrete strain $\epsilon(t)$ at time t is considered as a superposition of the following components:

$$\epsilon(t) = \epsilon^m(t) + \epsilon^n(t) \quad (B.1.1)$$

$$\epsilon^n(t) = \epsilon^c(t) + \epsilon^s(t) + \epsilon^a(t) + \epsilon^T(t) \quad (B.1.2)$$

where,

- $\epsilon(t)$ = total strain
- $\epsilon^m(t)$ = mechanical strain (the independent variable in the stress-strain relationship)
- $\epsilon^n(t)$ = nonmechanical strain composed of the following components:
 - $\epsilon^c(t)$ = creep strain
 - $\epsilon^s(t)$ = shrinkage strain
 - $\epsilon^a(t)$ = aging strain
 - $\epsilon^T(t)$ = temperature strain

The meanings of each of these strain components are discussed in this chapter and are illustrated for the case of a specimen under constant stress and no temperature change in **Figure B.1-1**.

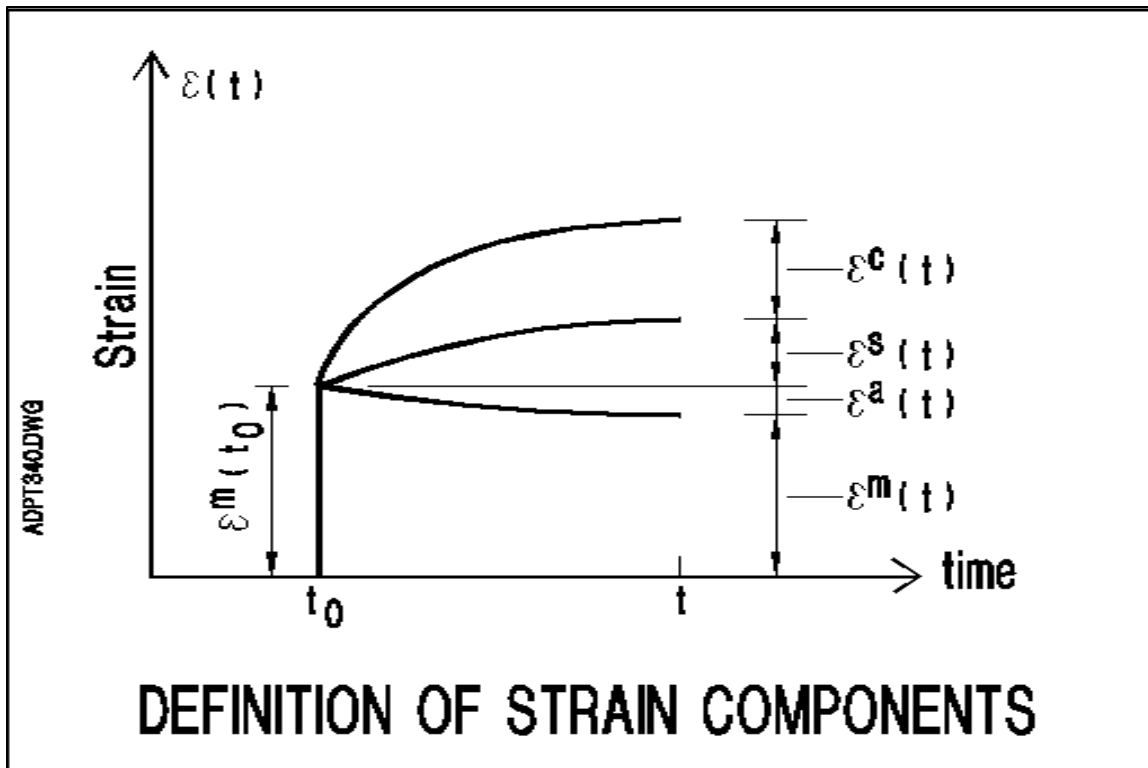


FIGURE B.1-1

B.1.1 Mechanical Strain

Mechanical strain $\epsilon^m(t)$ is a stress originated strain and is the independent variable in the concrete stress-strain relationship:

$$\sigma(t) = E(t) \epsilon^m(t) \quad (\text{B.1.3})$$

where $\sigma(t)$ is the uniaxial concrete stress at time t and $E(t)$ is the instantaneous elastic modulus at time t .

Neglecting any deterioration under high stress or cyclic loading, the elastic modulus $E(t)$ increases over time, quite rapidly during the first month after casting and more slowly afterwards. Thus under constant stress the mechanical strain $\epsilon^m(t)$ decreases over time.

The elastic modulus $E(t)$ is often estimated based on compressive strength. The compressive strength of concrete $f'_c(t)$ is defined as the maximum average stress obtained from the testing of concrete specimens such as cylinders, cubes, or prisms subjected to uniaxial compression.

B.1.2 Aging Strain

Aging strain $\epsilon^a(t)$ is a fictitious stress originated strain which can be defined as the decrease in mechanical strain over time due to the increase in elastic modulus of the concrete. Aging strain does not represent an actual physical deformation of the concrete and should instead be considered as a correction factor for the calculation of the current total stress as a function of the current total strain.

Under constant stress, the aging strain increment $\delta\epsilon^a$ occurring between time t_{n-1} and t_n may be expressed as:

$$\delta\epsilon^a = \sigma \left[\frac{1}{E(t_n - 1)} - \frac{1}{E(t_n)} \right] \quad (B.1.4)$$

B.1.3 Creep Strain

Creep strain $\epsilon^c(t)$ is a stress originated strain defined as the increase in strain under sustained stress of a concrete specimen under constant humidity and temperature. Creep strain does not include the instantaneous elastic deformation. Its rate decreases to zero over time, and is only partially recoverable after load removal.

The main factors influencing the creep of concrete are compressive strength, age at loading, aggregate type, ambient relative humidity and temperature, the specimen size, and the stress history. The most important factor from the standpoint of analysis is that creep strain depends on the entire stress history of the specimen. This makes it desirable to formulate a creep model which stores the stress history in a compact form.

Figure B.1-2 illustrates the relationship between instantaneous stress changes and the corresponding creep strain changes.

B.1.4 Shrinkage Strain

Shrinkage strain $\epsilon^s(t)$ is a non-stress originated strain defined as the deformation under no load or temperature change. Shrinkage of concrete is due primarily to loss of water upon drying (drying shrinkage) and volume change due to carbonation (carbonation shrinkage).

The main factors influencing the shrinkage of concrete are aggregate type, water-cement ratio, specimen size, and ambient relative humidity.

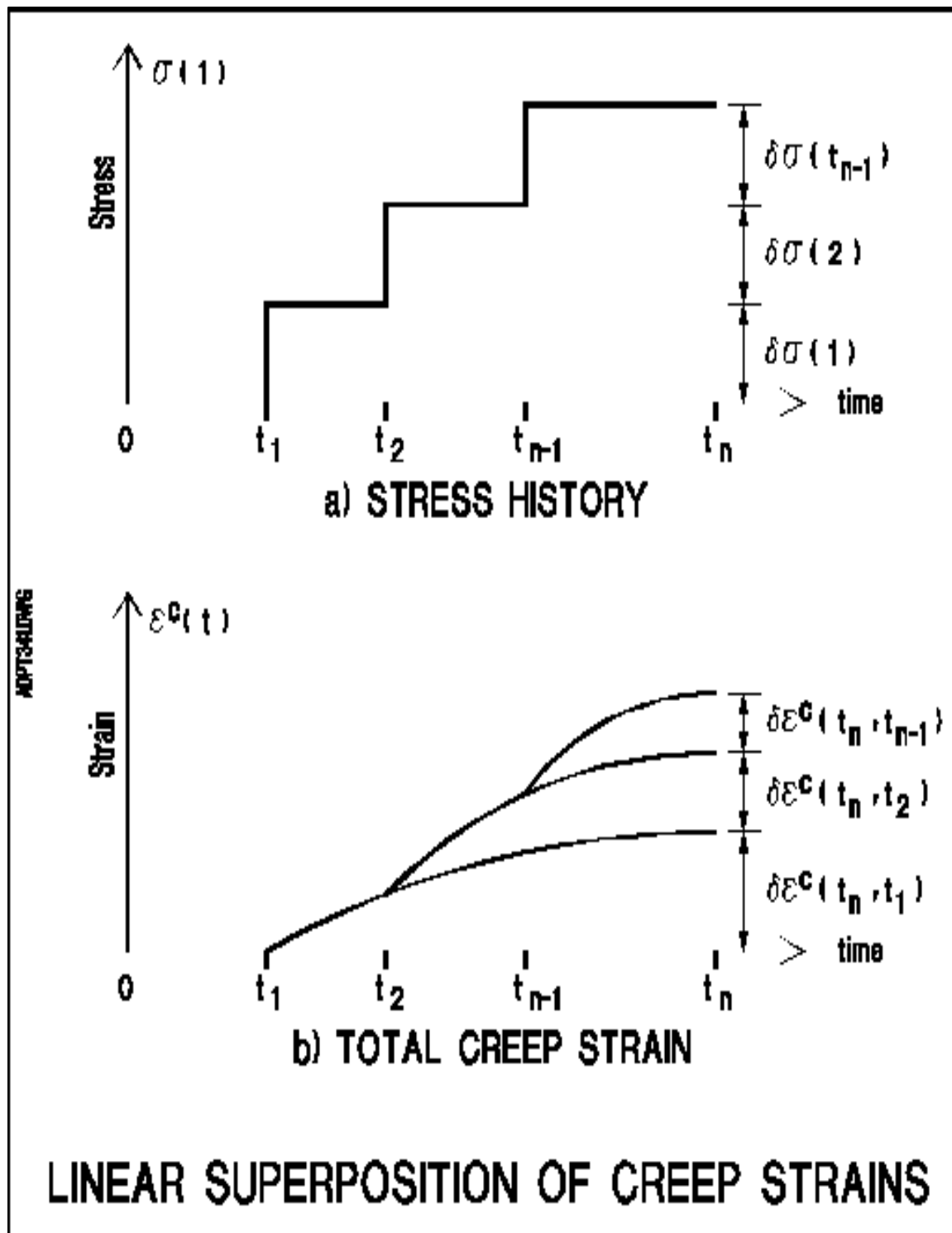


FIGURE B.1-2

B.1.5 Temperature Strain

Temperature strain $\epsilon^T(t)$ is a non-stress originated strain defined as the deformation under temperature change. The temperature strain increment $\delta\epsilon^T$ due to the temperature change δT may be expressed as $\delta\epsilon^T = \delta T \alpha$ where α is the coefficient of thermal expansion.

B.2 PREDICTION OF TIME DEPENDENT MATERIAL PROPERTIES

The experimental test data required for the analytical prediction of the time dependent strains in prestressed concrete structures are not always available. In this case the required material parameters can be estimated using empirical expressions based on mix design, humidity and other factors.

Several such empirical expressions for predicting the variation over time of concrete properties have been recommended by various researchers. Based on studies by Branson et al, ACI Committee 209 published a set of recommendations which have been widely applied in bridge design. An alternative approach adopted by the CEB - FIP recommendations (1978) is used in the bridge design examples published by the PCI - PTI. Both of these recommendations predict time dependent material parameters based on a few tests, estimates of cylinder properties, and environmental conditions.

Using procedures discussed in Section B.3 of this chapter, the stiffness, creep and shrinkage predictions found using one of these recommendations can be automatically incorporated into a time dependent analysis using the computer program described in Chapter 4.

The details of the assumptions and expressions of the recommendations are different and will not be discussed here. The relationships recommended for predicting the behavior of concrete structures may be summarized as follows:

B.2.1 ACI Committee 209 Recommendations

ACI Committee 209 recommendations provide a number of equations for predicting the time dependent material properties and strains in concrete.

A. Strength and stiffness

The cylinder strength $f'_c(t)$ is computed using an equation of the form

$$f'_c(t) = \frac{t}{a + bt} f'_c(28) \quad (B.2.1)$$

where $f'_c(28)$ is the 28-day strength, t is the time in days after casting of the concrete, and, a , and, b , are constants. The values of, a , and, b , depend on

the type of cement and the curing method used for the specimen. For type I cement and moist curing, the recommended values are $a = 4.0$ and $b = 0.85$. For constant f'_c over time $a = 0.0$ and $b = 1.0$.

The initial elastic modulus $E_i(t)$ is computed as a function of the cylinder strength $f'_c(t)$ using the following formula:

$$E_i(t) = 33 w^{1.5} [f'_c(t)]^{1/2} \quad (\text{B.2.2})$$

where w is the unit weight of the concrete in pounds per cubic foot and f'_c is the strength of the concrete in pounds per square inch.

B. Creep strain

Creep strain at any time under constant stress is computed using the following expression:

$$C(t) = K_s K_H K_h K_t \frac{(t - \tau)^{0.60}}{10 + (t - \tau)^{0.60}} C_u \quad (\text{B.2.3})$$

where,

$$C(t) = \text{creep coefficient} = \frac{\text{creep strain at any time } t}{\text{initial immediate strain}}$$

$$C_u = \text{ultimate creep coefficient determined by experiment.}$$

$$C_u = \frac{\text{creep strain at } \infty \text{ time after loading}}{\text{initial strain at time of loading}}$$

$$C_u = 2.35 \text{ for standard conditions}$$

$$K_s = \text{slump correction factor} \\ = 0.81 + 0.07s ; (s \text{ is in inches})$$

$$K_H = \text{humidity correction factor} \\ = 1.27 - 0.0067 H, \quad \text{when } H > 40 \%$$

$$H = \text{relative humidity}$$

$$K_h = \text{minimum thickness correction factor} \\ = 1.0 - 0.0167 (th - 6.0), \quad \text{when } th > 6 \text{ inches}$$

$$K_t = \text{age at loading correction factor} \\ = 1.25 \tau^{-0.118} \text{ for 7 days moist cured concrete}$$

$$s = \text{slump in inches}$$

$$th = \text{minimum size of member in inches}$$

$$t = \text{the age of concrete at observation time in days}$$

$$\tau = \text{the age of concrete at loading in days}$$

Standard conditions for creep, for which all correction factors are equal to 1.0 are as follows:

- (i) slump = 2.7 inches
- (ii) ambient relative humidity = 40 percent or less
- (iii) minimum thickness = 6 inches or less
- (iv) age at loading = 7 days for moist cured concrete

C. Shrinkage strain

Shrinkage strain is computed using the following relationship:

$$\epsilon^s(t) = \epsilon^s_u K_s K_h K_H \frac{(t - t_0)^e}{f + (t - t_0)^e} \quad (\text{B.2.4})$$

where,

- $\epsilon^s(t)$ = shrinkage strain at observation time, t
- ϵ^s_u = ultimate shrinkage strain, determined from experiments
- t = time at observation
- t_0 = age of curing
- f, e = constants, determined from experiments
- K_s = slump correction factor
- K_h = member size correction factor
- K_H = relative humidity correction factor

Normal ranges of constants e, f, and ϵ^s_u , using normal or light weight concrete for either moist curing or steam curing, are as follows:

- e = 0.90 to 1.10
- f = 20 to 130
- ϵ^s_u = 415×10^{-6} to 1070×10^{-6}

Standard relations can be selected for the prediction of shrinkage strains by selecting appropriate constants (ACI-209, 1982) for Equation B.2.4:

For concrete moist cured for 7 days:

$$\epsilon^s(t) = (800 \times 10^{-6}) K_s K_h K_H (t-7) / [35+(t-7)]$$

For concrete steam cured for 3 days:

$$\epsilon^s(t) = (730 \times 10^{-6}) K_s K_h K_H (t-3) / [55+(t-3)]$$

The correction factors are provided to take into account different field conditions that may exist for different cases under consideration.

- (i) Slump correction factor, K_s : Slump of the concrete mix is directly proportional to the water content of the mix. Hence, the less slump, the lower will be the amount of shrinkage. **Figure B.2-1(a)** shows the variation of the slump correction factor with slump.
- (ii) Humidity correction factor, K_H : Shrinkage decreases with increases in ambient humidity **Figure B.2-1(b)** shows the variation of the humidity correction factor with humidity.
- (iii) Size correction factor, K_h : Shrinkage decreases with increasing thickness of concrete members. **Figure B.2-1(c)** shows the variation of the size correction factor with thickness.

B.2.2 CEB - FIP Committee Recommendations

The CEB - FIP design recommendations include figures and tables for the estimation of time dependent material properties and strains in concrete. Unlike the ACI recommendations (1978), the CEB - FIP recommendations are not provided in the equation form required for computer analysis. However, Kristek and Smerda (1982) have proposed analytical approximations for these figures and tables. These approximations are suitable for computer analysis and are incorporated in the computer program described in this report.

A. Strength, modulus of elasticity and aging of concrete

- (i) The compressive strength $f'_c(t)$ is expressed as a function of time, based on the 28- day compressive strength $f'_c(28)$.
- (ii) The initial elastic modulus is estimated as a function of time and the compressive strength using the formula

$$E_i(t) = 45680 [f'_c(t)]^{1/2} \quad (\text{B.2.5a})$$

where,

$E_c(t)$ = modulus of elasticity on day t, in N/cm².

$f'_c(t)$ = concrete strength on day t, in N/cm².

- (iii) The change in modulus of elasticity with time is effected using the following relationship:

$$f'_c(t) = [(1.45 \cdot t^{0.75}) / (t^{0.75} + 5.5)] \cdot f'_c(28) \quad (\text{B.2.5b})$$

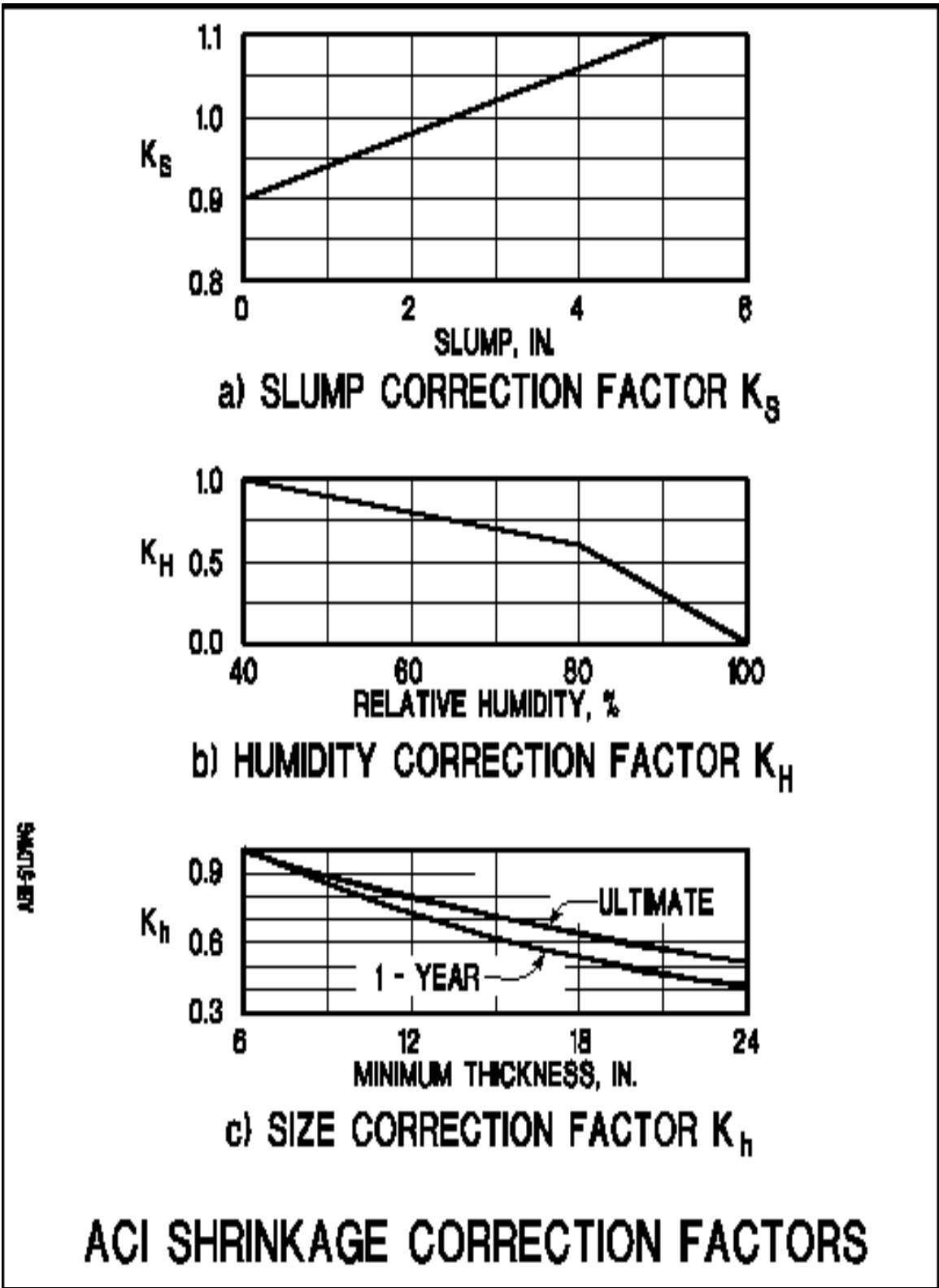


FIGURE B.2-1

where,

$f_c(t)$ = concrete strength on day t .

$f_c(28)$ = concrete strength on day 28.

B. Creep strain

The creep coefficient ϕ is and defined as the ratio of creep strain to elastic strain based on the 28-day elastic modulus:

$$\epsilon^c(t, \tau) = \sigma_0 \phi(t, \tau) / E(28) \quad (B.2.6)$$

where,

$\epsilon^c(t, \tau)$ = creep strain at time t under a stress σ_0 applied at time τ

$\phi(t, \tau)$ = creep coefficient for age of loading τ and observation time t

t = the age of the concrete at observation time.

τ = the age of the concrete when the load is applied.

The creep coefficient $\phi(t, \tau)$ under working conditions is evaluated as the sum of three components:

$$\phi(t, \tau) = \beta_a(\tau) + \phi_d \beta_d(t - \tau) + \phi_f [\beta_f(t) - \beta_f(\tau)] \quad (B.2.7)$$

where,

$$\beta_a(\tau) = 0.8 [1 - f_c(\tau) / f_c(\infty)] \quad (B.2.8)$$

ϕ_d = the delayed modulus of elasticity, taken as 0.4

ϕ_f = the flow coefficient. $\phi_f = \phi_{f1} \cdot \phi_{f2}$

ϕ_{f1} = depends on the humidity (Table B.1-1).

ϕ_{f2} = depends on the notional thickness.

β_d = is a function for the development over time of the delayed elastic strain.

β_f = a function for the development over time of the delayed plastic (flow) strain.

$\beta_a(\tau)$ represents the irreversible part of the creep strain which takes place during the first few days after application of the load.

$\phi_d \beta_d(t - \tau)$ represents the recoverable delayed elastic part of the creep strain which is independent of aging.

$\phi_f [\beta_f(t) - \beta_f(\tau)]$ represents the irreversible flow part of the creep strain which is influenced by the age of the concrete at loading.

The factors ϕ_{f2} and $\beta_f(t)$ are functions of the notional thickness h_0 of the concrete elements. The notional thickness is defined by

$$h_0 = \Gamma^2 A_c / \mu \quad (\text{B.2.9})$$

where,

Γ = notional thickness - humidity factor (Table B.1-1).

A_c = area of concrete cross section (in cm^2).

μ = length of the perimeter of the concrete cross section in contact with the atmosphere (in cm).

TABLE B.1-1 BASIC COEFFICIENTS OF CREEP AND SHRINKAGE

Ambient Environment	Relative humidity	Coefficient of creep j_{f1}	Coefficient of shrinkage e_{s1}	Coefficient G
1	2	3	4	5
Water	-	0.8	+0.00010	30
Very damp atmosphere	90%	1.0	-0.00013	5
Outside in general	70%	2.0	-0.00032	1.5
Very dry atmosphere	40%	3.0	-0.00052	1

The age of the concrete is adjusted to account for variations in temperature and the type of cement used. For each period δt_m during which the mean temperature is T_m , the corrected age is obtained from the expression:

$$t = \alpha \sum [T_m + 10] \delta t_m / 30 \quad (\text{B.2.10})$$

where,

t = corrected age

α = 1.0 for ASTM cement types I and II

α = 2.0 for ASTM cement type III

α = 3.0 for high early strength cement types

T_m = ambient temperature ($^{\circ}\text{C}$) during δt_m days

δt_m = number of days with temperature T_m

When concrete cures at 20°C (68°F) and normal (ASTM type I) cement is used, the corrected age and the real age are equivalent.

C. Shrinkage strain

The shrinkage strain over the time interval (t-τ) is given by

$$\epsilon^s(t, \tau) = \epsilon_{su} [\beta_s(t) - \beta_s(\tau)] \quad (B.2.11)$$

where,

- ϵ_{su} = the ultimate shrinkage strain. $\epsilon_{su} = \epsilon_{s1} \cdot \epsilon_{s2}$
 ϵ_{s1} depends on the humidity (Table B.1-1).
 ϵ_{s2} depends on the notional thickness.
- β_s = a function for the development of shrinkage strain over time
which also depends on the notional thickness.
- t = the age of the concrete at observation time.
- τ = the age of the concrete at the beginning of the time interval.

The notional thickness (Equation B.2.9) and the corrected age (Equation B.2.10) defined for the creep model are also used for this shrinkage model.

B.3 MATHEMATICAL MODELING OF THE CREEP STRAIN COMPONENT

Creep of concrete presents one of the most complex numerical problems in the time dependent analysis of concrete structures. The modeling of creep is a statistical problem involving curve fitting, and the many influential factors discussed above are involved. The most important factor from the standpoint of formulating a mathematical model is that the creep strain depends on the entire stress history of the specimen. This makes it desirable to formulate a creep model which stores the stress history in a compact form.

B.3.1 Theoretical Background

Creep strain may be expressed in terms of the following convolution integral for an aging viscoelastic material:

$$\epsilon^c(t) = \int_0^t J(t, \tau) \frac{d\sigma}{d\tau} d\tau \quad (B.3.1)$$

where $\epsilon^c(t)$ is the creep strain at time t, and $J(t, \tau)$ is the specific creep compliance function for observation time t under initial loading at time τ. The inherent assumptions behind this formulation are:

- A. the applicability of the principle of superposition of strain components (Equation B.1.1); and
- B. a linear relationship between creep strain and applied stress.

The form of this creep strain relationship is convenient for a step-by-step displacement method finite element analysis, but its direct evaluation is impractical because it requires integration over the entire history of stresses in an element. This can require a tremendous numerical effort and data storage capacity for the solution of realistic problems.

The need for storing and using the complete history of stresses can be eliminated by approximating the specific creep compliance function $J(t, \tau)$, also known as the kernel, with a so-called degenerate kernel. This basic approach has been developed by many researchers including Bresler and Selna (1964), Zienkiewicz and Watson (1966), Mukaddam (1969), Kabir and Scordelis (1976), Bazant and Wu (1973) and others. Some of these researchers also include the instantaneous elastic strain in the kernel. Bazant and Wittmann (1982) thoroughly analyze this formulation and also discuss convergence properties of alternative degenerate kernels. The most general degenerate kernel takes the form of a Dirichlet series which can be written as:

$$J(t, \tau) = \sum_{i=1}^m a_i(\tau) [1 - e^{(y_i(\tau) - y_i(t))}] \quad (B.3.2)$$

In this investigation $y_i(t) = t/\Gamma_i$ is used, the instantaneous elastic strain is not included in the kernel and the degenerate kernel takes the form used by Kabir (1976):

$$J(t, \tau) = \sum_{i=1}^m a_i(\tau) [1 - e^{-(t-\tau)/\Gamma_i}] \quad (B.3.3)$$

where $a_i(\tau)$ are creep compliance coefficients which depend on the age of loading τ , and Γ_i are retardation times in days which govern the shape of the creep curve. Kabir also included a temperature shift function which is not included in this formulation. This degenerate kernel can be made to approximate any creep compliance function $J(t, \tau)$ to any degree of accuracy by selecting the number of terms m and the coefficients $a_i(\tau)$ and Γ_i on the basis of least squares curve fitting. This curve fitting procedure is discussed in Section B.3.3 of this Chapter.

Use of this kernel will result in any one of several different expressions for computing creep strain, depending on the assumed variation of stresses and material parameters over the length of a time step. Assuming step function stress variations and constant material parameters results in the expression given by Kabir

(1976) (Equation B.3.11). Assuming linear stress variations and constant material parameters results in an expression similar to one given by Bazant (1982) (Equation B.3.14). Assuming linear stress variations and linear variations of material parameters results in yet another expression (Equation B.3.19).

The recursive relationships for computing the creep strain may be derived as follows. Substituting the degenerate kernel (Equation B.3.2) into the convolution integral (Equation B.3.1), the resulting expression for $\epsilon^c(t)$ may be written as:

$$\epsilon^c(t) = \int_0^t \sum_{i=1}^m a_i(\tau) \left[1 - e^{(y_i(\tau) - y_i(t))} \right] \frac{d\sigma(\tau)}{dy_i(\tau)} \frac{dy_i(\tau)}{d\tau} d\tau \quad (B.3.4)$$

Rearranging and simplifying this expression results in an equation for $\epsilon^c(t)$ in the form:

$$\epsilon^c(t) = \sum_{i=1}^m \epsilon_i(t) \quad (B.3.5)$$

in which,

$$\epsilon_i(t) = \int_0^t a_i(\tau) \frac{d\sigma(\tau)}{dy_i(\tau)} \frac{dy_i(\tau)}{d\tau} d\tau - g_i(\tau) \quad (B.3.6a)$$

and,

$$g_i(t) = e^{-y_i(t)} \int_0^t a_i(\tau) e^{y_i(\tau)} \frac{d\sigma(\tau)}{dy_i(\tau)} \frac{dy_i(\tau)}{d\tau} d\tau \quad (B.3.6b)$$

Expressing the derivatives $d\epsilon_i/dy_i$ and $d^2\epsilon_i/dy_i^2$, it can be verified by substitution into Equation B.3.6 that $\epsilon_i(t)$ always satisfy the following linear differential equations:

$$\frac{d^2 \epsilon_i}{dy_i^2} + \frac{d \epsilon_i}{dy_i} = a_i(\tau) \frac{d\sigma}{dy_i} \frac{dy_i}{d\tau} d\tau \quad (B.3.7)$$

Expressing the derivative dg_i/dy_i from Equation B.3.6, it can be verified that $g_i(t)$ always satisfy the following linear differential equations:

$$\frac{dg_i}{dy_i} + g_i = a_i(\tau) \frac{d\sigma}{dy_i} \frac{dy_i}{d\tau} d\tau \quad (B.3.8)$$

The derivatives with respect to y_i may be expressed in terms of time derivatives through the following substitutions:

$$\begin{aligned} d\epsilon_i/dy_i &= \dot{\epsilon}_i/\dot{y}_i \text{ and} \\ d^2\epsilon_i/dy_i^2 &= (\ddot{\epsilon}_i \dot{y}_i - \dot{\epsilon}_i \ddot{y}_i) / \dot{y}_i^3 \end{aligned}$$

Making this substitution in equation B.3.7 results in the following differential equation:

$$\ddot{\epsilon}_i + \left(\dot{y}_i - \frac{\ddot{y}_i}{\dot{y}_i} \right) \dot{\epsilon}_i = a_i(t) \dot{y}_i \dot{\sigma} \quad (\text{B.3.9})$$

This type of constitutive relationship corresponds to a Kelvin chain rheological model consisting of aging springs and dashpots.

B.3.2 Calculation of the Creep Strain Increment

The recursive relationships for calculating the increment in creep strain over one time step can be derived from these differential equations based on any one of a number of different assumptions regarding the variation of stresses and material parameters over the length of the time step. The relationships for the following sets of assumptions are derived and discussed below:

- A. Constant stress and constant material parameters over each time step;
- B. Linear variation in stress and constant material parameters over each time step;
- C. Linear variation in stress and linear variation in material parameters over each time step.

The recursive relationships resulting from each of these sets of assumptions are described below and are implemented in the computer program described in Chapter 4.

To describe a time step, let time be subdivided by discrete times t_j ($j=1,2,3,\dots$) and let δ refer to increments from t_{j-1} to t_j . Then:

$$\delta y_i = y_i(t_j) - y_i(t_{j-1})$$

$$\delta \sigma = \sigma(t_j) - \sigma(t_{j-1})$$

etc.

A. Constant stress and constant material parameters

Assuming $a_i(t)$ and $\sigma(t)$ to be constant over the time step from t_{j-1} to t_j , and with a step function $\delta \sigma$ in $\sigma(t)$ at time t_{j-1} , the integration of Equation B.3.8 then yields exactly, for uniaxial stress

$$g_i(t_j) = g_i(t_{j-1}) e^{-\delta y_i} + a_i(t_{j-1}) \delta \sigma \quad (\text{B.3.10})$$

By substituting Equation B.3.10 into Equation B.3.6b and taking $y_i(t) = t/\Gamma_i$, Equation B.3.5 and Equation B.3.6a may be rewritten, resulting in the following recursive relationship for evaluation of the creep strain increment over the time step

$$\delta \epsilon^c = \sum_{i=1}^m g_i(t_{j-1}) [1 - e^{-\delta t/\Gamma_i}] \quad (\text{B.3.11a})$$

$$g_i(t_j) = g_i(t_{j-1}) e^{-\delta t/\Gamma_i} + a_i(t_{j-1}) \delta \sigma_{j-1} \quad (\text{B.3.11b})$$

$$g_i(t_0) = 0 \quad (\text{B.3.11c})$$

where g_i are called the hidden state variables, and are updated during each time step for re-use in the next time step. This relationship for evaluating the creep strain increment over the time step is equivalent to the relationship used by Kabir (1976), although its derivation is different.

B. Linear stress and constant material parameters

Assuming $a_i(t)$ and $d\sigma(t)/dy_i$ to be constant over the time step from t_{j-1} to t_j , and setting $d\sigma(t)/dy_i = \delta \sigma / \delta y_i$, the integration of Equation B.3.8 then yields exactly, for uniaxial stress

$$g_i(t_j) = g_i(t_{j-1}) e^{-\delta y_i} + a_i(t_{j-1}) \Phi_i \delta \sigma \quad (\text{B.3.12})$$

$$\Phi_i = [1 - e^{-\delta y_i}] / \delta y_i \quad (\text{B.3.13})$$

By substituting Equation B.3.12 into Equation B.3.6b and taking $y_i(t) = t/\Gamma_i$, Equation B.3.5 and Equation B.3.6a may be rewritten, resulting in the following recursive relationship for evaluation of the creep strain increment over the time step

$$\delta \epsilon^c = R \delta \sigma + \sum_{i=1}^m g_i(t_{j-1}) [1 - e^{-\delta t/\Gamma_i}] \quad (\text{B.3.14a})$$

$$R = \sum_{i=1}^m a_i(t_{j-1}) (1 - \Phi_i) \quad (\text{B.3.14b})$$

$$\Phi_i = \Gamma_i [1 - e^{-\delta t/\Gamma_i}] / \delta t \quad (\text{B.3.14c})$$

$$g_i(t_j) = g_i(t_{j-1}) e^{-\delta t/\Gamma_i} + a_i(t_{j-1}) \Phi_i \delta \sigma \quad (\text{B.3.14d})$$

$$g_i(t_0) = 0 \quad (\text{B.3.14e})$$

Because the creep strain increment over the time step, $\delta \epsilon^c$, is dependent on the stress change over the time step, $\delta \sigma$, in Equation B.3.14a, this relationship

must be evaluated iteratively in each time step. All terms in Equation B.3.14a other than $\delta\sigma$ stay the same for all iterations, therefore all the summations can be done prior to starting the iterations, and only $R \delta\sigma$ needs evaluation in each iteration. The iteration can stop when $\delta\sigma$ undergoes little change from one iteration to the next.

The time increment δt appears in the denominator of Equation B.3.14c above. Thus the recursion is impossible to evaluate when, $\delta t=0$. Investigation of limits as δt approaches zero provides the following expression for use in place of Equation B.3.14c when, $\delta t=0$.

$$\Phi_i = 1 \quad (B.3.15)$$

C. Linear stress and linear material parameters

Assuming $da_i(t)/dy_i$ and $d\sigma(t)/dy_i$ to be constant over the time step from, t_{j-1} , to, t_j , and setting, $da_i(t)/dy_i = \delta a_i / \delta y_i$, and $d\sigma(t)/dy_i = \delta\sigma / \delta y_i$, the integration of Equation B.3.8 then yields exactly, for uniaxial stress

$$g_i(t_j) = g_i(t_{j-1}) e^{-\delta y_i} + [a_i(t_{j-1}) \Phi_i + \beta_i(t_{j-1}) (1-\Phi_i)] \delta\sigma \quad (B.3.16)$$

$$\Phi_i = [1 - e^{-\delta y_i}] / \delta y_i \quad (B.3.17)$$

$$\beta_i = \delta a_i / \delta y_i \quad (B.3.18)$$

By substituting Equation B.3.16 into Equation B.3.6b and taking $y_i(t) = t/\Gamma_i$, Equation B.3.5 and Equation B.3.6a may be rewritten, resulting in the following recursive relationship for evaluation of the creep strain increment over the time step

$$\delta \epsilon^c = R \delta\sigma + \sum_{i=1}^m g_i(t_{j-1}) [1 - e^{-\delta t / \Gamma_i}] \quad (B.3.19a)$$

$$R = \sum_{i=1}^m a_i(t_{j-1}) (1-\Phi_i) + \beta_i(t_j) [(\delta t / 2\Gamma_i) - (1-\Phi_i)] \quad (B.3.19b)$$

$$\Phi_i = \Gamma_i [1 - e^{-\delta t / \Gamma_i}] / \delta t \quad (B.3.19c)$$

$$\beta_i = \Gamma_i \delta a_i / \delta t \quad (B.3.19d)$$

$$g_i(t_j) = g_i(t_{j-1}) e^{-\delta t / \Gamma_i} + [a_i(t_{j-1}) \Phi_i + \beta_i(t_{j-1}) (1-\Phi_i)] \delta\sigma \quad (B.3.19e)$$

$$g_i(t_0) = 0 \quad (B.3.19f)$$

Because the creep strain increment over the time step $\delta\epsilon^c$ is dependent on the stress change over the time step $\delta\sigma$ in Equation B.3.19a, this relationship must be evaluated iteratively in each time step. All terms in Equation B.3.19a other than $\delta\sigma$ stay the same for all iterations, therefore all the summations can be done prior to starting the iterations, and only $R \delta\sigma$ needs evaluation in each iteration. The iteration can stop when $\delta\sigma$ undergoes little change from one iteration to the next.

The time increment δt appears in the denominators of Equation B.3.19c and B.3.19d above. Thus the recursion is impossible to evaluate when $\delta t = 0$. Investigation of limits as δt approaches zero provides the following expressions for use in place of Equation B.3.19c and Equation B.3.19d when $\delta t = 0$.

$$\Phi_I = 1 \quad (B.3.20)$$

$$\beta_I = 0 \quad (B.3.21)$$

B.3.3 Determination of creep compliance coefficients

The material properties for creep strain computation in this algorithm are the creep compliance coefficients $a_i(t)$ and retardation times Γ_i . They must be available as data to the subroutines implementing the creep strain algorithm. Kabir (1976) demonstrated how they can be evaluated from creep data from an experimental study, and also used the method to find coefficients corresponding to the ACI Committee 209 (1982) recommendations.

The method of evaluating the coefficients is the same whether the creep strains are measured in a laboratory or computed using some formula for their estimation such as those discussed in Chapter B.2. The *test* data required for the concrete are the values of total creep strain under unit stress $J(t, \tau)$ for a number of loading ages τ and observation times t . Then the following procedure is followed in order to make a best fit approximation of the test data with the degenerate kernel (Equation B.3.3):

1. m and Γ_i , $i=1, m$ are chosen on a trial basis.
2. A particular age τ_0 is chosen.
3. Various times t_j , $j=1, 2, \dots, n$ are chosen such that $t_j > \tau_0$.
4. Values of $J(t_j, \tau_0)$ are found at $j=1, 2, \dots, n$ points.
5. The following system of equations is set up:

$$\begin{bmatrix} 1 - e^{-(t_1 - \tau_0)/\Gamma_1} & 1 - e^{-(t_1 - \tau_0)/\Gamma_2} & \dots & 1 - e^{-(t_1 - \tau_0)/\Gamma_m} \\ 1 - e^{-(t_2 - \tau_0)/\Gamma_1} & 1 - e^{-(t_2 - \tau_0)/\Gamma_2} & \dots & 1 - e^{-(t_2 - \tau_0)/\Gamma_m} \\ \vdots & \vdots & & \vdots \\ 1 - e^{-(t_n - \tau_0)/\Gamma_1} & 1 - e^{-(t_n - \tau_0)/\Gamma_2} & \dots & 1 - e^{-(t_n - \tau_0)/\Gamma_m} \end{bmatrix} \begin{bmatrix} a_1(\tau_0) \\ a_2(\tau_0) \\ \vdots \\ a_m(\tau_0) \end{bmatrix} = \begin{bmatrix} J(t_1, \tau_0) \\ J(t_2, \tau_0) \\ \vdots \\ J(t_n, \tau_0) \end{bmatrix}$$

or in condensed form

$$[A]_{n \times m} [a]_{m \times 1} = [b]_{n \times 1}, \quad n > m \quad (B.3.14)$$

6. The method of least squares is applied to solve the overdeterminate system of equations for the values of $[a]$.

$$[A]^T_{m \times n} [A]_{n \times m} [a]_{m \times 1} = [A]^T_{m \times n} [b]_{n \times 1}$$

Solution of Equation B.3.14 for the unknown vector $[a]$ gives the classical least squares solution

$$[a]_{m \times 1} = [A^T A]^{-1}_{m \times m} [A^T b]_{m \times 1}$$

7. Choose a different m and Γ_i and go through steps 2 through 6, for a new $[a]$
8. Optimum m and Γ_i 's are chosen based on the following criteria:
 - (a) Least square errors are minimized.
 - (b) $\sum a_i(\tau)$ which is the ultimate creep strain closely matches the ultimate creep strain.
 - (c) The contributions of all $a_i 1 - e^{-(t - \tau)/\Gamma_i}$ terms are approximately equal.
9. Choose a different age τ_1 and repeat steps 3 through 6 to determine a new set of $a_i(\tau_1)$. This is repeated for several values of τ to provide a table of $a_i(\tau)$ for different loading ages τ . Coefficients for different loading ages may be determined by interpolation between values given in the table.

This approach may be applied to experimental data or to data generated by some empirical relationship for predicting creep strain.

For the CEB - FIP creep model discussed in Section B.2.2, this approach must be applied twice; once for the total, recoverable plus irreversible, creep and once for the recoverable creep component only. Then when the creep strain increment $\delta \epsilon^c$ and the hidden state variables g_i are evaluated in each time step, the coefficients for total creep are used when the creep strain

increment over the time step and the total creep strain are of the same sign, and the coefficients for the recoverable component only are used when the creep strain increment over the time step and the total creep strain are of different signs.

This Page Left Intentionally

BLANK

C. REFERENCES

The following references are consulted or used in the preparation of this manual:

Aalami, B. O., (1993), *Analysis and Design of Segmentally Constructed Bridges*, Proceedings, International Conference on Concrete Engineering and Technology, Kuala Lumpur, Institution of Engineers, Malaysia, May 25-27, 1993, pp. TS2-1-16

AASHTO, (1991), *Interim Specifications - Bridges -1991*

AASHTO, (1989a), *Guide Specifications for Design and Construction of Segmental Concrete Bridges*.

AASHTO, (1989b), *Standard Specifications for Highway Bridges*, 14th edition.

Abbas, S., and Scordelis, A. C., (1990), *Non-linear Analysis of Segmentally Erected Reinforced and Prestressed Concrete Cable-Stayed Bridges*, Proceedings, Second Workshop on Bridge Engineering Research in Progress, University of Nevada, Reno, Nevada, Oct 29-30, 1990, pp.11-14

Abdel-Karim, A. M., and Tadros, M. K., (1992), *Design and Construction of Spliced I-Girder Bridges*, PCI Journal, July-August 92, pp. 114-122.

ACI-209, (1982), *Prediction of Creep, Shrinkage and Temperature Effects in Concrete Structures*, Designing for Creep and Shrinkage in Concrete Structure, ACI Publication SP-76, American Concrete Institute, Detroit, pp. 193-300.

Bazant, Z. P., and Wittman, F. H., (1982), *Creep and Shrinkage in Concrete Structures*, Book, New York, John Wiley and Sons.

Bazant, Z. P., and Wu, S. T., (1973), *Dirichlet Series Creep Function for Aging Concrete*, Journal of Engineering Mechanics Division, ASCE, V.99, No. EM2, Paper 9645, pp. 367-378.

Breen, J. E., Cooper, R. L., and Gallaway, T. M., (1975), *Minimizing Construction Problems in Segmentally Precast Box Girder Bridges*, Research Report No. 121-6F, Center for Highway Research, The University of Texas at Austin, Texas.

Bresler, B. and Selna, L. G., (1964) *Analysis of Time-Dependent Behavior of Reinforced Concrete Structures*, ACI Symposium on Creep of Concrete, American Concrete Institute, SP-9, detroit.

- Brown, R. C., Burn, N. H., Breen, J. E., (1974), *Computer Analysis of Segmentally Erected Precast Box Girder Bridges*, Research Report No. 121-4, Center for Highway Research, The University of Texas at Austin, Texas.
- CALTRANS, (1988), *Bridge Design Practice Manual* California Department of Transportation, Sacramento, Ca.
- CEB, (1978), *CEB-FIP Model-Code for Concrete Structures*, Bulletin d'Information, 124/125E, Comite' Euro-International du Beton, Federation Internationale de la Precontrainte, Paris, 348 pp.
- Collins, M. P., and Mitchel, D., (1990), *Prestressed Concrete Structures*, Prentice Hall, NJ, pp. 776.
- FDOT, (1989a), *Post-Tensioning Manual*, Florida Department of Transportation, Office of Construction, October 1989.
- FDOT, (1989b), *Segmental Manual*, Florida Department of Transportation, Office of Construction, October 1989.
- Hernandez, H. D. and Gamble E. L., *Time Dependent Prestress Losses in Pretensioned Concrete Construction*, Structural Research Series No. 417, Civil Engineering Studies, University of Illinois, Urbana, May 1975.
- Kabir, A. F., (1976), *Nonlinear Analysis of Reinforced Concrete Panels, Slabs and Loading*, University of California at Berkeley, SESM Report No 76-6.
- Kang, Y. J., and Scordelis, A. C., (1990), *Non-linear Segmental Analysis of Reinforced and Prestressed Concrete Bridges*, Proceedings, Third International Conference on Short and Medium Span Bridges, Toronto, Ontario, Canada, Vol 1, pp. 229-240.
- Ketchum, M. A. and Scordelis, A. C., (1986), *Redistribution of Stresses in Segmentally Erected Prestressed Concrete Bridges*, University of California, Berkeley, Report No. UCB/SESM-86-07.
- Kristek, V., and Smerda, Z., (1982), *Simplified Calculation of the Relaxation of Stress Respecting the Delayed Elasticity*, Fundamental Research on Creep and Shrinkage of Concrete, Edited by Wittman, H. F., Martinus Nijhoff Publishers, Boston.
- McGregor, James (1988), *Reinforced Concrete, Mechanics and Design*, Prentice Hall, New Jersey, 800 pp.

Lacy, G. C. and Breen, J. E. (1975), *The Design and Optimization of Segmentally Precast Prestressed Box Girder Bridges*, Research Report No. 121-3, Center for Highway Research, The University of Texas at Austin, Texas, 1975.

Lacy, G. C. and Breen, J. E. (1969), *Long Span Prestressed Concrete Bridges of Segmental Construction*, Research Report No. 121-1, Center for Highway Research, The University of Texas at Austin, Texas, 1975.

Libby, J. R., (1976), *Segmental Box Girder Bridge Superstructure Design*, ACI Journal, May 1976, No. 12-23, pp. 279-290.

Lin, C.S., (1973), *Non-linear Analysis of Reinforced Concrete Slabs and Shells*, University of California, Berkeley, UCB-SESM 73-7

Magura, D. D., Sozen, M. A. and Siess, C. P. *A Study of Stress Relaxation in Prestressing Reinforcement*, PCI Journal, Vol. 9, No. 2, April 1964.

Mukaddam, M. A., (1969), *Behavior of Concrete under Variable Temperature and Loading*, Interim Report to Oak Ridge National Laboratory, Reactor Division, Oak Ridge, Tennessee.

NCHRP, (1992), *Development of Comprehensive Bridge Specifications and Commentary*, National Cooperative Highway Research Program, Project 12-33 Report.

PTI, (1985), *Post-Tensioning Manual*, Post-tensioning Institute, Phoenix, AZ, 406 pp.

PTI, (1990), *Post-Tensioned Box Girder Bridge Manual*, Post-tensioning Institute, Phoenix, AZ, 146 pp.

Scordelis, (1993), *Ketchum Bridge - Comparison of Linear Analysis Results*, Solution table received through private correspondence from Professor Scordelis, University of California, Berkeley, July 1993.

Scordelis, A. C., Chan, E. C., Ketchum, M. A., and Van Der Walt, P. P., (1985), *Computer Programs for Prestressed Concrete Box Girder Bridges*, University of California Publication, SESM-85/02, 346 pp.

Van Zyl, S. F., (1978), *Analysis of Curved Segmentally Erected Prestressed Concrete Box Girder Bridges*, University of California, Berkeley, UCB SESM 78-2, Jan 1978, pp.265

VSL, (1992), *H3 Viaduct in Hawaii*, shop drawings, VSL Corporation, Los Gatos, Ca, USA.

Wilson, E.L., and Hoit, M.I., (1984), *A Computer Adaptive Language for the Development of Structural Analysis Programs*, Computers and Structures, Vol. 19, No. 3, pp 321-338, 1984.

Zienkiewicz, O. C., and Watson, M., (1966), *Some Creep Effects in Stress Analysis with Particular Reference to Concrete Pressure Vessels*, Nuclear Engineering and Design, No. 4.

D. BACKGROUND TO CAMBER COMPUTATION

D.1 INTRODUCTION.....3

D.2 NUMERICAL EXAMPLE.....5

 D.2.1 Input data for ADAPT-ABI 9

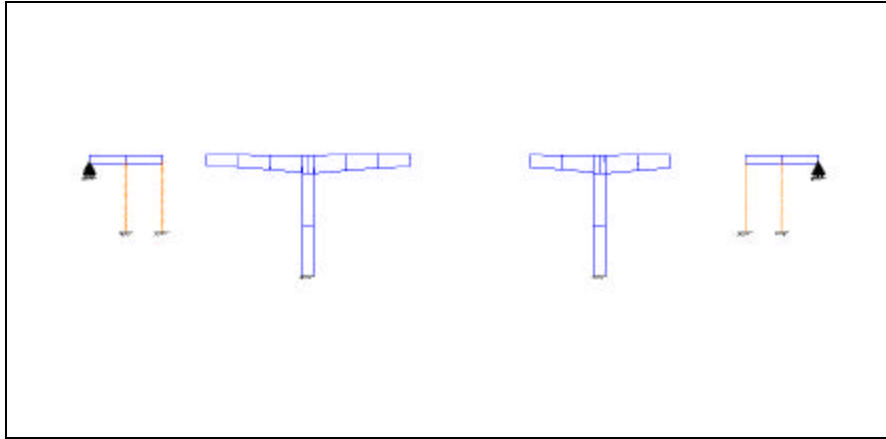
This Page Left Intentionally

BLANK

D.1 INTRODUCTION

Camber is defined as the offset from a datum line built into a structure at time of construction. The offset is generally a distance above the horizontal line (datum). The purpose of the camber is to compensate for the deflection of the structure under loading, and due to time dependent effects. The compensation is aimed at bringing the final position of the structure to within acceptable limits for the structure's intended function. The amount of camber is selected to offset the entire, or part of the anticipated deflection of a structure.

Camber computation for incrementally constructed bridges, in particular balanced cantilever construction (**Figure D.1-1**) is a critical step in the bridge design. Since, not only the two tips of the approaching cantilevers must closely meet at the closure, the in-service long-time profile of the bridge must be smooth and in a position determined by design.



VIEW OF A TYPICAL CANTILEVER CONSTRUCTION BRIDGE
FIGURE D.1-1

This writing illustrates the basics of camber computation for balanced cantilever construction. It concludes with a numerical example using ADAPT-ABI.

Consider **Figure D.1-2**. It shows a cantilever consisting of four segments, built in four stages. At each stage, one segment is added to the structure. Each time a new segment is added, its weight increases the deflection of the previously installed segments. For example, the deflection of node 2 shown for stage 1 increases with the progress of construction. The completed structure, shown for stage 4, will have a downward deflection as indicated in the illustration.

The first step in camber calculation is to determine where (at which height with respect to datum line) each of the nodes (common face of two adjacent segments) should be installed during construction, so that the completed structure would end up on the datum line (horizontal line). Since the deflection of the cantilever is downward, it is apparent that the

nodes must be constructed above the datum line in anticipation that the deflection of the structure will lower them to the level of datum line.

In bridge structures, it is generally desired that the structure would conclude with a profile above the datum line. The second step in camber calculation is, therefore, to add to the camber values from the previous step, the amount of the desired upward offset from the horizontal line.

As an example, let us review the camber calculation for node 4 of the cantilever in **Figure D.1-2** for which the camber necessary to bring the final position of the bridge to the horizontal line is determined. Since in the first step of computation the camber shall fully eliminate the anticipated deflection along the bridge, the magnitude of camber must be:

$$\text{Camber} = \text{Initial position} - \text{Final position}$$

Initial position is the location (height) where a node (segment end) is originally placed at time of construction. In this example, node 4 is at the right end of segment 3, and it is installed at point A. Its position with respect to the datum line at time of installation is $d_{4,i}$. The suffix indicates location of node 4, at the initial condition. The basis for computation of the initial position of node 4 will be expounded further on. After completion of the construction and lapse of time, node 4 will end up in its **final position** having a computed displacement equal to $d_{4,4}$. This is the displacement of node 4 at stage 4.

Hence, the camber for this node is:

$$\text{Camber}_4 = d_{4,i} - d_{4,4}$$

In regards to the initial position of node 4, the following consideration applies. Refer to **Figure D.1-2** stage 2.5. It is assumed that the form traveler is positioned as an extension of segment 2, ready to receive the new segment 3. The form traveler, shown in the figure with a straight broken line, has the same slope at node 3 as segment 2. Hence, the initial position of node 4 on the form, and prior to placement of concrete is:

$$d_{4,i} = d_{3,2} + \theta_{3,2} * L_3$$

Where,

$d_{3,2}$ = the displacement of node 3 at stage 2. This displacement already includes the effects from the weight of the form traveler.

$\theta_{3,2}$ = the rotation of node 3 at stage 2.

Once concrete is placed, the form traveler would deflect. Point A (stage 2.5) will move below its initial position by an amount controlled by the stiffness of the form traveler. Once

concrete is set and post-tensioning is applied, segment 3 lifts off the formwork. Point 4 assumes a new position. Hence, the final position of point 4 depends largely on the stiffness of the form traveler (deflection of the forms and the traveler truss under selfweight of concrete), and the amount and time of prestressing. Once segment 3 is stripped, the form traveler is moved and positioned to cast segment 4.

In summary, the final deflection of the construction and the camber calculation are very sensitive to the stiffness details of the form traveler, prestressing and the construction sequence. The ADAPT-ABI allows for these effects in its deflection and camber calculations. The outcome of the computations, however, would be as accurate as the input data justifies.

D.2 NUMERICAL EXAMPLE

For illustration purposes, a cantilever with dimensions given in the input data presented herein is selected. For simplicity and without compromising the concept of computations, neither prestressing nor a traveler is included in the example. The omission of form traveler and prestressing would require that each segment be several days old at time of installation (refer to input data for details).

The construction of the cantilever is shown in **Figure D.1-2**. The time sequence for installation of segments is:

Stage 1	Day 10
Stage 2	Day 25
Stage 3	Day 40
Stage 4	Day 60

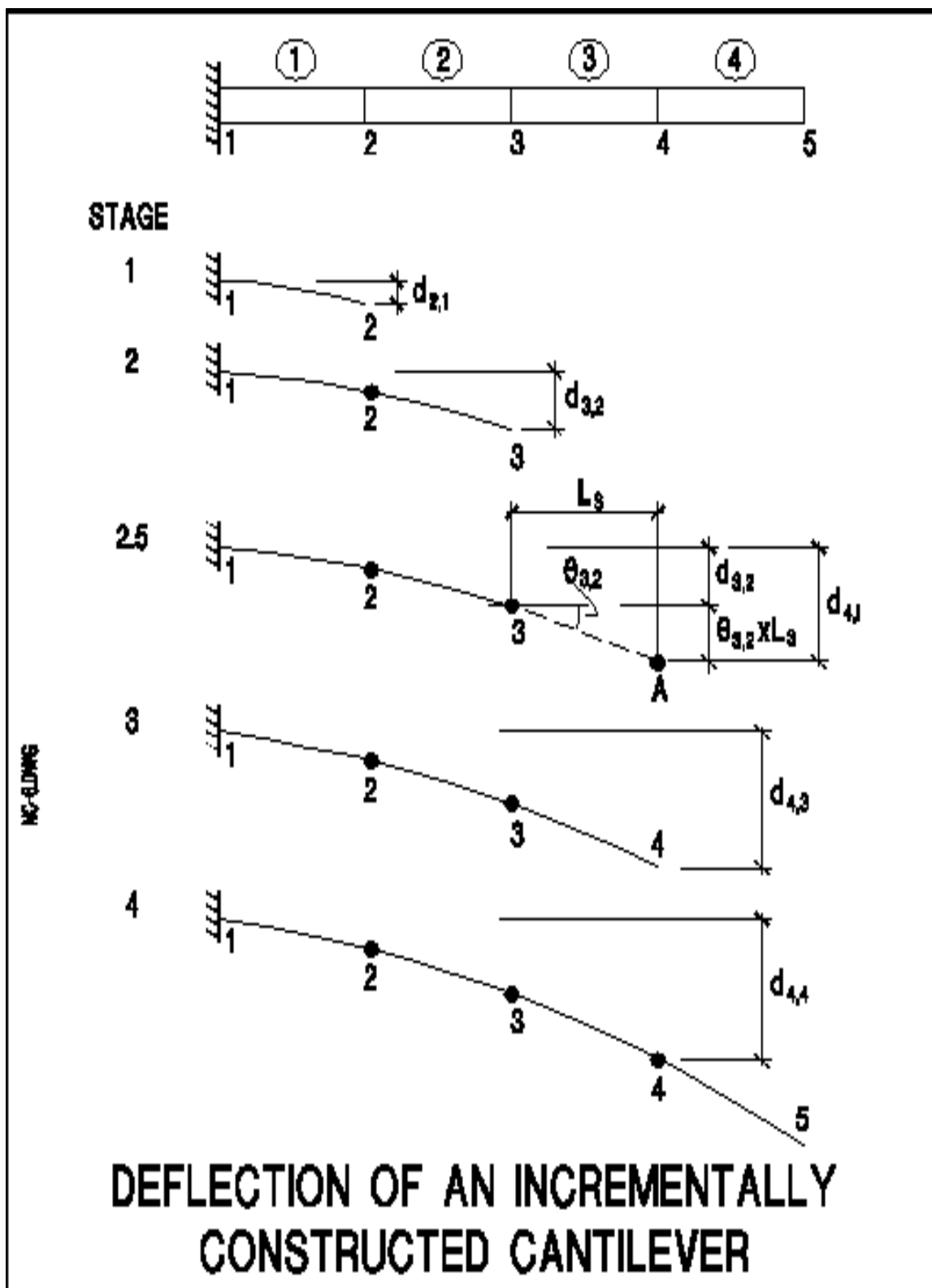


FIGURE D.1-2

The solution for the displacement of the nodes at each stage and the camber are reproduced from the ADAPT-ABI output in the following.

STAGE 1 - NODAL TOTAL DISPLACEMENTS

NODE	U(X)	U(Y)	R(Z)	X-DISP	Y-DISP	Z-ROTN
1	1	1	1	0.0000E+00	0.0000E+00	0.0000E+00
2	0	0	0	-1.8925E-02	-3.5619E-02	-5.3428E-05

STAGE 2 - NODAL TOTAL DISPLACEMENTS

NODE	U(X)	U(Y)	R(Z)	X-DISP	Y-DISP	Z-ROTN
1	1	1	1	0.0000E+00	0.0000E+00	0.0000E+00
2	0	0	0	-8.9188E-02	-1.4024E-01	-2.3215E-04
3	0	0	0	-1.3540E-01	-4.0959E-01	-2.8796E-04

STAGE 3 - NODAL TOTAL DISPLACEMENTS

NODE	U(X)	U(Y)	R(Z)	X-DISP	Y-DISP	Z-ROTN
1	1	1	1	0.0000E+00	0.0000E+00	0.0000E+00
2	0	0	0	-1.3180E-01	-3.1258E-01	-5.4898E-04
3	0	0	0	-2.3677E-01	-9.9670E-01	-7.7225E-04
4	0	0	0	-2.8298E-01	-1.8062E+00	-8.2806E-04

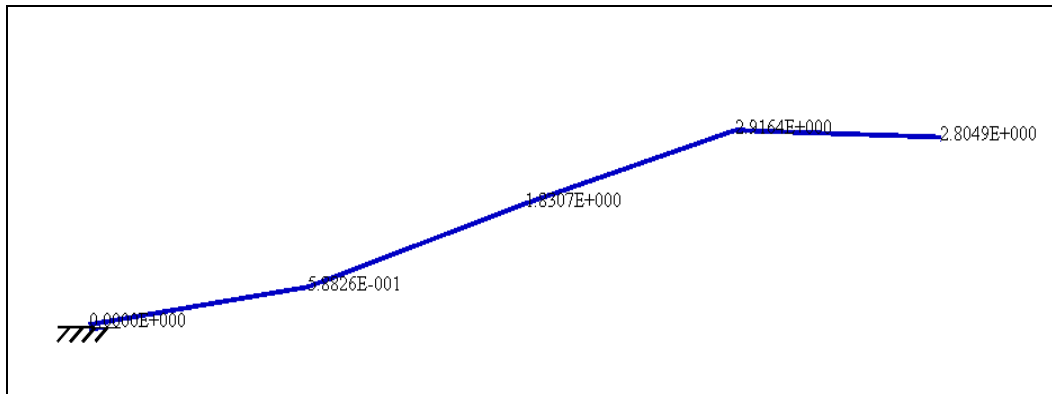
STAGE 4 - NODAL TOTAL DISPLACEMENTS

NODE	U(X)	U(Y)	R(Z)	X-DISP	Y-DISP	Z-ROTN
1	1	1	1	0.0000E+00	0.0000E+00	0.0000E+00
2	0	0	0	-1.6900E-01	-5.6346E-01	-1.0220E-03
3	0	0	0	-3.2102E-01	-1.8946E+00	-1.5649E-03
4	0	0	0	-4.4077E-01	-3.5980E+00	-1.7938E-03
5	0	0	0	-5.0956E-01	-5.4299E+00	-1.8510E-03

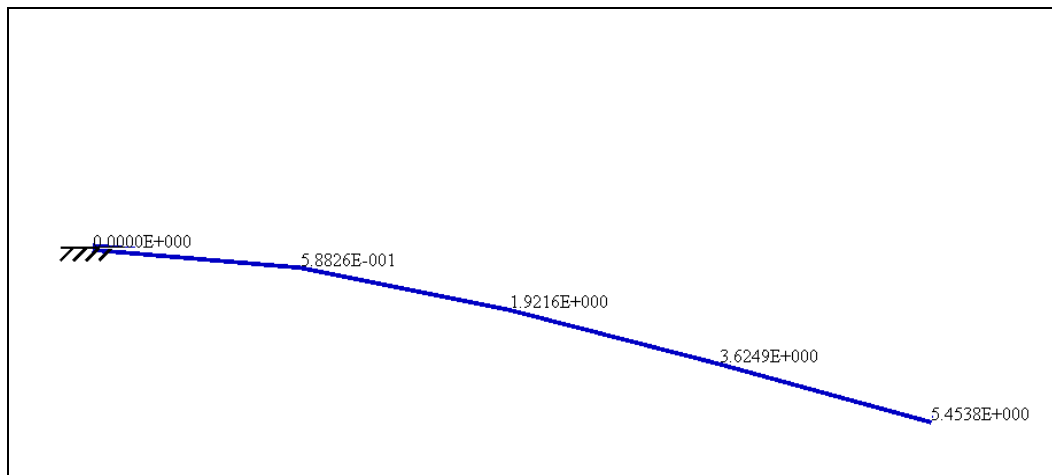
CAMBER

NODE	X-DIRECTION	Y-DIRECTION	ROTATION
1	0.0000E+00	0.0000E+00	0.0000E+00
2	0.1690E+00	0.5635E+00	0.1022E-02
3	0.3021E+00	0.1806E+01	0.1511E-02
4	0.3054E+00	0.2900E+01	0.1506E-02
5	0.2266E+00	0.2796E+01	0.1023E-02

The deflected shape and the required camber for the completed structure is shown in **Figure D.2-1** from ADAPT-ABI graphical output. The following is the long-hand calculation for camber at node 4 (shown as 1.18) in the figure. Refer to the **Figure D.1-2** for the symbols used.



(a) Deflected Profile



(b) Camber

ILLUSTRATION OF DEFLECTION AND CAMBER OF AN
INCREMENTALLY CONSTRUCTED CANTILEVER
FIGURE D.2-1

$$d_{3,2} = -0.40959 \quad (\text{deflection of node 3 at stage 2})$$

$$\theta_{3,2} = -2.8796\text{E-}4 \quad (\text{rotation of note 3 at stage 3})$$

$$d_{4,i} = d_{3,2} + \theta_{3,2} * L_3 \quad (\text{initial position of node 4, at installation})$$

$$= -0.40959 - 2.8796\text{E-}4 * 1000 = -0.6976$$

$$d_{4,4} = -3.5980 \quad (\text{final position of node 4 at completion of structure - stage 4})$$

$$\text{Camber 4} = -0.6976 + 3.5980 = 2.900 \quad (\text{software output is 2.900 OK})$$

Note that in **Figure D.2-1(b)**, for practical considerations, the camber shown is the vertical displacement of each node. But, the displacement in **Figure D.2-1** are the vectorial sum of displacements in the horizontal and vertical directions (along x- and y-axis of the frame). For example, the displacement 3.62 shown in **Figure D.2-1** for node 4 is $(3.598^2 + 0.4407^2)^{1/2}$.

D.2.1 Input data for ADAPT-ABI

```

=====
;          ADAPT-BRIDGE Incremental  ABI  Software Manual Example
;          =====
;          Name of this file : CAMBER1.INP
;          Units: N,mm
; This example illustrates the application of camber computation
;
START
TITLE N=2
      ABI CAMBER COMPUTATION EXAMPLE
      COMPUTE CAMBER FOR A SEGMENTALLY CONSTRUCTED CANTILEVER
UNITS U=SI

CONCRETE PARAMETERS N=1
  1  M=CEB1
MESH INPUT
  NODES N=5
    1  X=0      Y=0
    5  X=4000   Y=0   G=1,5

  CONCRETE PROPERTIES N=1
    1  Fpc=34  Cr=2.5  Sh=0.0004  W=2.4E-6
  MILD STEEL PROPERTIES N=1
    1  Es=200000  P=0.02
  SECTION PROPERTIES N=1
    1  B=300  D=500
  ELEMENTS N=4
    FRAME N=4
      1,1,2  C=1  X=1  ST=1  Day=0
      2,2,3  C=1  X=1  ST=1  Day=15
      3,3,4  C=1  X=1  ST=1  Day=30
      4,4,5  C=1  X=1  ST=1  Day=45

MESH COMPLETE

SET Day=10
CHANGE STRUCTURE
  BUILD N=1                      ; Insllall the first segment
  RESTRAINTS
    1 R=1,1,1

CHANGE COMPLETE
SOLVE Day=10 ! OUTPUT           ; Obtain solution at day=10

CHANGE STRUCTURE
  BUILD N=2

```

```
CHANGE COMPLETE
SOLVE
SOLVE Day=25 ! OUTPUT
```

```
CHANGE STRUCTURE
  BUILD N=3
CHANGE COMPLETE
SOLVE
SOLVE Day=40 ! OUTPUT
```

```
CHANGE STRUCTURE
  BUILD N=4
CHANGE COMPLETE
SOLVE
SOLVE Day=60 ! OUTPUT
```

```
CAMBER                                ; This command triggers a printout of camber
                                      ; computations in the output
STOP
```